

Green Kenuue

Reference Manual

September 2010



Canadian Hydraulics Centre
National Research Council
Ottawa, Ontario, Canada



Water Survey Canada
Environment Canada
Ottawa, Ontario, Canada

Copyright ©1998-2010 Canadian Hydraulics Centre, National Research Council

Green Kenue was developed with Microsoft© Visual C++, Copyright ©1994-2010, Microsoft Corporation. All Rights Reserved.

MFC. Microsoft Corporation. Copyright ©1997. All Rights Reserved.

OpenGL. Silicon Graphics, Inc. Copyright ©1993. All Rights Reserved.

WATFLOOD/SPL9, Copyright © by N. Kouwen, 1986-2000

Supported Foreign Files:

TOPAZ, Grazinglands Research Laboratory and Department of Geography, University of Saskatchewan.

ArcInfo® Grid, Environmental Systems Research Institute, Inc. Copyright ©1997-2002.

ArcView® Shape Files, Environmental Systems Research Institute, Inc. Copyright ©1997-2002.

DTED® or CDED® DEM, National Imagery and Mapping Agency (NIMA).

MapInfo® Interchange Format, MapInfo Corporation. Copyright ©2002.

Surfer® Grid, Golden Software, Inc. Copyright ©1994-1997.

GeoTIFF Library, Copyright ©1995, Niles D. Ritter. Copyright ©1999, Frank Warmerdam.

TIFF Library, Copyright ©1988-1997, Sam Leffler. Copyright ©1991-1997, Silicon Graphics, Inc.

ACKNOWLEDGEMENTS

Green Kenue has been a collaborative effort funded in part by:

- Environment Canada, Ottawa, Ontario

Special thanks are directed at several individuals who have supported the Green Kenue project by serving as beta testers and providing technical feedback, constructive criticism, and helpful comments:

- Al Pietroniro, National Hydrology Research Centre, Environment Canada, Saskatoon, Saskatchewan
- Jean-Guy Zakrevsky, Water Survey Canada, Environment Canada, Ottawa, Ontario
- Stuart Hamilton, Pacific Yukon Region, Meteorological Service of Canada, Environment Canada, Vancouver, British Columbia
- David Hutchinson, Pacific Yukon Region, Meteorological Service of Canada, Environment Canada, Vancouver, British Columbia
- Nicholas Kouwen, Department of Civil Engineering, University of Waterloo, Waterloo, Ontario
- David Morin, Environmental Protection Services, Environment Canada, Gatineau, Quebec
- Erika Klyszejko, Water Survey Canada, Environment Canada, Ottawa, Ontario
- Raymond Bourdages, Technical Development, Water Survey Branch, Environment Canada, Ottawa, Ontario
- Maurice Sydor, Data Integration Modelling and Analysis, Environment Canada, Gatineau, Quebec

Thanks are also given to the many users who have provided feedback and suggestions for other applications within the EnSim family.

Table of Contents

1 ENSIM CORE	1
1.1 A QUICK OVERVIEW	1
1.1.1 The EnSim Simulation Environment	1
1.1.2 Getting Started	1
1.1.3 Getting Help with EnSim	2
1.1.3.1 Conventions in EnSim Help	2
1.2 THE WORKSPACE	3
1.2.1 Managing Objects in the WorkSpace	3
1.2.2 Saving and Loading The WorkSpace	6
• To Save a WorkSpace:	6
• To Load a WorkSpace:	6
1.3 THE ENSIM INTERFACE	7
1.3.1 The Menu Bar	7
1.3.2 The Tool Bar	8
1.3.3 The T3 Mesh Editing Toolbar	8
1.3.4 Shortcut Menus	8
1.4 DATA ITEMS	9
1.4.1 Loading and Importing Data Items	10
1.4.1.1 Native Data Items	10
1.4.1.2 Foreign Data Items	11
1.4.2 Saving and Exporting Data Items	11
1.4.3 Properties of Data Items	17
1.4.3.1 Display Properties	17
1.4.3.1.1 Rendering Options	18
1.4.3.1.2 Vertical Display Options	19
1.4.3.1.3 Other Display Options	19
1.4.3.2 Colour Scale	20
• To edit the colour scale:	20
• To apply a previously created colour scale:	21
1.4.3.3 Data Attributes	21
1.4.3.4 Spatial	25
1.4.3.4.1 Attributes	26
1.4.3.4.2 Coordinate Systems	26
1.4.3.4.3 Coordinate System - Converting Projections	27
• To change the projection of the object:	27
1.4.3.4.4 Coordinate Systems - Assigning Projections	27
• To assign a coordinate system to an object:	27
1.4.3.4.5 Ellipsoids	28
1.4.3.4.6 Selecting a Coordinate System	29
1.4.3.5 Meta Data	31

1.4.3.6 Applying Changes to an Object's Properties	31
1.4.3.7 Copying Data Item Properties	31
• To copy data item properties:	31
1.5 VIEWS	33
1.5.1 Creating a View Window	33
1.5.2 Removing a View Window	33
1.5.3 Properties Shared by all View Types	33
1.5.3.1 The Properties Dialog	34
1.5.4 The 1D View Window	34
1.5.4.1 Labels of Axes in a 1D View	35
1.5.4.2 The 1D View Window Status Bar	35
1.5.4.3 Manipulating the 1D View	36
1.5.4.4 Display Properties of the 1D View Window	36
1.5.5 The Polar View Window	37
1.5.5.1 Coordinates in a Polar View	38
1.5.5.2 The Polar View Window Status Bar	38
1.5.5.3 Manipulating the Polar View	38
1.5.5.4 Display Properties of the Polar View Window	39
1.5.6 The 2D View Window	40
1.5.6.1 Coordinate Systems and Units in 2D Views	40
1.5.6.2 The 2D Window Status Bar	41
1.5.6.3 Manipulating the 2D View	41
• To move a data item already in the view to the top layer:	41
1.5.6.4 Display Properties of the 2D Window	42
1.5.7 The 3D View Window	43
1.5.7.1 The 3D Window Status Bar	43
1.5.7.2 Manipulating the 3D View	44
1.5.7.3 Display Properties of the 3D View Window	45
1.5.8 The Spherical View Window	46
1.5.8.1 The Spherical View Window Status Bar	47
1.5.8.2 Manipulating the Spherical View	47
1.5.8.3 Display Properties of the Spherical View Window	48
1.5.9 The Report View Window	49
1.5.9.1 The Report View Window Status Bar	49
1.5.9.2 The Report View Window Tool Bar	50
1.5.9.3 Manipulating the Report View	51
• To add a view to a report:	51
• To manipulate a view that has been added to a report:	51
• To change the order of objects in the report:	51
• To change the border around an object in the report:	51
1.5.9.4 Report View Window Page Setup Properties	52
1.5.9.5 Report View Templates	53
• To create a report template:	53
• To use a report template:	53
1.5.10 View Decorations	53

Table of Contents September 2010

1.5.10.1 Legends	54
1.5.10.1.1 Colour Scale Legends	54
1.5.10.1.2 Independent Legends	56
• To create an independent legend:	56
• To edit an independent legend:	56
• Creating a Quick Legend:	57
1.5.10.2 The Compass	58
1.5.10.3 The Simulation Clock	58
1.5.10.4 Labels	59
• To create a label:	59
• To edit a label:	60
1.5.11 Displaying Base Maps	60
• To Display a Base Map:	60
1.5.12 Animation	61
1.5.13 Flight Paths	62
• To create a new flight path:	62
1.5.13.1 Flight Path Properties	63
1.5.14 Synchronizing Two Views	63
• To synchronize Views:	64
1.5.15 Saving and Copying Images	65
1.5.15.1 Recording	65
• To create a movie:	65
1.5.15.2 Copying to the Clipboard	66
• To copy the image of a view window to the clipboard:	66
1.5.15.3 Printing	66
1.5.16 Troubleshooting in Views	66
1.6 TOOLS	68
1.6.1 Creating New Data Items	68
1.6.1.1 Drawing Points	68
• To create a point set:	68
1.6.1.2 Drawing Lines and Closed Polylines	69
• To create a line or polyline:	69
• To create a closed line or polygon:	69
1.6.1.3 Creating a New Regular Grid	70
• To create a new Regular Grid:	70
1.6.1.4 Creating a New Triangular Mesh	71
• To create a new Triangular Mesh from existing data:	71
1.6.1.5 Creating a New Table Object	72
• To create a new Table Object from existing data:	72
1.6.2 Selecting Data Items	73
1.6.3 Editing Data Items	73
1.6.3.1 Editing Attributes	73
• To edit an attribute of a data object:	73
1.6.3.2 Editing Points	75
• To edit a point within a point set:	75
• To add a point:	75
• To delete a point:	75

1.6.3.3	Editing Lines	76
	• To edit the value of a line:	76
	• To edit the location of a point within a line set:	76
1.6.3.3.1	Transferring Lines within Line Sets	76
	• To transfer a line from one Line Set to another:	76
	• To duplicate a line in two or more Line Sets:	77
1.6.3.3.2	Editing Line Segments	77
	• To append a line segment to another line segment:	77
	• To turn a Closed Line into an Open Line:	77
	• To divide a Closed or Open Line into two Open Lines:	78
1.6.3.3.3	Adding Line Segments	78
	• To add an Open Line to a Line Set:	78
	• To add a Closed Line to a Line Set:	78
1.6.3.4	Editing T3 Meshes	79
1.6.3.5	Resampling Data	80
1.6.3.5.1	Resampling Lines and LineSets	80
	• To Resample a Line or LineSet:	80
1.6.3.6	Shifting Data Objects	82
	• To relocate a Data Object:	82
1.6.4	Probing Data	82
1.6.4.1	Data Probes	83
	• To probe data:	83
1.6.4.2	The Live Cursor	85
1.6.4.2.1	The Live Stream Lines Cursor	86
	• To save stream lines:	87
1.6.4.3	The Ruler	87
1.6.4.4	Computing Areas and Volumes	88
	• To compute an area:	88
	• To compute a volume:	89
1.6.5	Extracting Data	89
1.6.5.1	Extracting Surfaces	90
1.6.5.1.1	Extracting Temporal Statistics	90
	• To extract temporal statistics as a surface:	90
1.6.5.1.2	Extracting Slopes	91
1.6.5.1.3	Extracting Aspects	92
1.6.5.1.4	Extracting Curvatures	93
1.6.5.2	Extracting Residuals	95
1.6.5.3	Extracting Isolines	95
	• To extract an isoline:	95
1.6.5.4	Extracting Paths	95
	• To extract a path:	96
1.6.5.5	Extracting Points	96
	• To extract points from a data item:	96
1.6.5.6	Extracting Time Series	96
	• To extract a time series:	96
1.6.5.7	Extracting a Velocity Rose	99
	• To extract a velocity rose:	99
1.6.5.8	Extracting an Attribute Table	100
	• To Extract an Attribute Table	100
1.6.5.9	Extracting Data From a Mesh	101

1.6.5.9.1	Extracting a Subset of a Mesh	101
•	To extract a subset of a mesh:	101
1.6.5.9.2	Extracting the Edge of a Mesh	102
•	To extract the edge of a mesh:	102
1.6.5.9.3	Extracting Edge Lengths From a Mesh	102
•	To extract the edge lengths of a mesh:	102
1.6.5.10	Extracting Integrals	103
•	To extract an integral along a line	103
1.6.5.11	Extracting XY Data From a Table	103
•	To extract XY data from a table:	103
1.6.6	TimeSeries Tools	104
1.6.6.1	Editing Time Series	104
1.6.6.2	Resampling Time Series	107
1.6.6.3	Computing Performance Statistics	109
•	To compute performance statistics:	109
1.6.6.4	Computing a Flow Duration Curve	110
•	To compute a flow duration curve:	110
1.6.6.5	Computing a Cumulative Sum	110
•	To compute a cumulative sum:	110
1.6.6.6	Computing an Integral	110
•	To compute the integral of a TimeSeries:	111
1.6.6.7	Computing a Distribution	111
•	To create a distribution:	111
1.6.7	Create Vector Field	112
•	To create a vector grid or mesh:	112
1.6.8	Mapping Objects	113
•	To map objects:	114
1.6.9	Calculators	115
1.6.9.1	The Calculator for Data Items	115
•	To use the calculator:	115
1.6.9.2	The Calculator for Gridded Objects	116
•	To use the calculator:	116
1.6.9.3	The Calculator for Time Series Objects	118
•	To use the calculator:	118
1.6.9.4	The Calculator Expressions	120
1.7	How To - HINTS AND TRICKS	122
1.7.1	Draping a 3D Image Onto a DEM	122
•	To drape an image:	122
1.7.2	Extracting Cross-Sections from Gridded Data	123
•	To create a cross-section or 3D polyline:	123
1.7.3	Extracting Cross-Sections from Points and Line Data	124
•	To create a cross-section or 3D polyline from points or line data:	124
1.7.4	Displaying Two Features of an Object Simultaneously	124
1.7.5	Displaying Isoline-Outlined Filled Contours	125
1.7.6	Creating a Sloping Structure in a Rectangular Grid	126
1.7.7	Extracting a Spatial Subset From a Larger Grid	127
•	To define a spatial subset from a rectangular grid:	127

1.7.8 Extracting a Temporal Subset of Time-Varying Gridded Data	127
• To extract a temporal subset:	128
1.7.9 Digitizing from an Imported Image	128
• To digitize from an imported image:	128
1.7.10 Georeferencing a non-georeferenced GeoTIFF	129
• To georeference a non-georeferenced tiff:	129
1.7.11 Classification of a GeoTIFF Image	130
• To classify a GeoTIFF image:	130
• To create a Custom Theme:	131
• To choose from a predefined theme:	131
• To reclassify a GeoTIFF	132
2 GREEN KENUE	135
2.1 WATERSHED OBJECTS	135
2.1.1 Opening an Existing Watershed Object	136
2.1.2 Importing a Watershed from Topaz	136
2.1.3 Creating a New Watershed Object	136
2.1.3.1 Watersheds	137
2.1.3.1.1 Flow Algorithms	138
2.1.3.1.2 Delineating a Watershed	138
• To delineate a watershed:	138
2.1.3.1.3 Delineating a Multi-Tile Watershed	139
• To delineate a multi-tile watershed:	139
2.1.3.2 DEMs	140
2.1.3.2.1 Checking for Errors and Editing the DEM	141
2.1.3.3 Channels and Flow Paths	141
2.1.3.3.1 Channel Attributes	142
2.1.3.3.2 Displaying Channels	143
• To view more or fewer channels:	144
2.1.3.3.3 Editing the Channels	145
2.1.3.3.4 Using Predefined Channels	145
• To add a predefined channel to a Watershed:	147
2.1.3.3.5 Watershed or Basin Outlet Nodes	147
• To select an outlet node:	148
• To select a channel node near a watershed outlet node:	148
2.1.3.4 Basin or Watershed Boundaries	149
2.1.3.4.1 Creating and Removing Basins	150
• To add a basin:	150
• To remove a basin:	150
2.2 HYDROLOGIC TOOLS	151
2.2.1 Watershed Tools	151
2.2.1.1 Extracting Drainage Directions	151
2.2.1.2 Extracting Drainage Area	152
2.2.1.3 Extracting Depression Fill	153
2.2.1.4 Extracting Average Upslope Elevation	154
2.2.1.5 Extracting Average Upslope Slope	155
2.2.1.6 Extracting Wetness Index	155
2.2.1.7 Extracting Stream Power	156

2.2.1.8 Extracting Relief Potential	157
2.2.1.9 Extracting Upstream Network	157
2.2.1.10 Extracting Downstream Reach	158
2.2.1.11 Extracting Basin Network	159
2.2.1.12 Extracting a Hypsographic Curve	159
2.2.1.13 Extracting Basin Flow Path Distances	160
2.2.1.14 Drainage Area Ratio Analysis	161
• To launch the DAR Analysis:	161
2.2.1.14.1 Known Flow	162
• To add a known flow station:	162
• To remove a known flow station:	162
2.2.1.14.2 Computed Flow	163
• To add a computed flow station:	163
• To remove a known flow station:	164
2.2.1.15 Slope Analysis	164
• To launch the Slope Analysis Dialog:	164
2.2.2 Rating Curve Analysis (RCA)	165
2.2.2.1 Background and Theory	165
2.2.2.1.1 Power Curve Fit	165
2.2.2.1.2 Polynomial Curve Fit	166
2.2.2.2 The Rating Curve Analysis Interface	166
• To perform an RCA on a HYDAT station:	166
• To create an RCA from any two time series:	166
2.2.2.2.1 Working With Rating Curves	169
• To adjust the rating curve by creating a subset:	169
• To inactivate an individual data point:	169
• To adjust the rating curve directly:	170
2.2.2.3 Opening an Existing RCA	174
• To open an RCA:	174
2.2.2.4 Saving an RCA	174
• To save an RCA:	174
2.3 WATFLOOD	175
2.3.1 WATFLOOD Map Files	175
2.3.1.1 Opening an Existing Watflood Map File	175
2.3.1.2 Creating a New Watflood Map File	175
• To set map file specifications manually:	175
• To set map file specifications automatically:	176
• To return to the default grid:	177
2.3.1.3 Modelling Multiple Watersheds	178
2.3.1.4 Watflood Map Data Attributes	179
2.3.1.4.1 Description of Data Attributes	179
2.3.1.4.2 Calculating the Default Data Attributes from the Watershed Object	182
2.3.1.4.3 Displaying Different Data Attributes in the Watflood Map	182
• To display the same data attributes for all cells:	182
• To display all the data attributes for a single cell:	183
2.3.1.5 Editing Watflood Map Data Attributes	183
2.3.1.5.1 Adding Land Use Data Using Closed Polygons	183
• To add land class data:	184
• To map land use data:	184

• Points to remember when applying land use data to a Watflood Map:	186
2.3.1.5.2 Adding Land Use Data Using GeoTIFFs	186
• To map land use data:	186
• Points to remember when creating land use classes from GeoTIFF images:	187
2.3.1.5.3 Editing Land Use Data	187
2.3.1.5.4 Resetting a Land Use Class	187
• To reset a land use class:	187
2.3.1.6 Saving the Watflood Map	187
2.3.2 Importing WATFLOOD Files	187
2.3.2.1 Watflood Event File Properties	188
• To save changes to the Event file:	189
2.3.3 WATFLOOD Output	189
2.3.4 Bankfull Animation	190
• To create a bankfull animation:	190
3 ENVIRONMENTAL DATABASES	193
3.1 HYDAT DATABASE	193
3.1.1 Introduction	193
3.1.2 Accessing the Database	193
3.1.2.1 The HYDAT Database CD	193
• To access the HYDAT database CD:	193
3.1.2.2 The HYDAT MDB Database	194
• To open the HYDAT MDB Database:	194
3.1.3 Accessing Station Details	195
• To access a selected station:	195
• To access a station by ID:	196
3.1.4 Filtering Station Details	196
• To filter the HYDAT MDB Database:	196
3.1.5 Properties of a HYDAT Station	198
3.1.5.1 Details	198
3.1.5.2 HYDEX	199
3.1.5.3 Meta Data	200
3.1.6 Properties of Associated Time Series	201
3.1.6.1 Subset	201
• To create a temporal subset:	201
3.2 CDCD DATABASE	203
3.2.1 Introduction	203
3.2.2 Accessing the Database	203
• To access the CDCD database:	203
3.2.3 Accessing Station Details	204
• To access a selected station:	204
• To access a station by ID:	205
3.2.4 Properties of a CDCD Station	205
3.2.4.1 Details	205
3.2.4.2 Meta Data	206
3.2.5 Properties of Associated Time Series	206

3.2.5.1	Subset	207
3.3	NARR DATABASE	209
3.3.1	Introduction	209
3.3.2	Downloading the NARR Data	209
3.3.3	Accessing the NARR Variables	210
• To import the NARR data:		210
4	THE GEN1D MODEL	215
4.1	GENERAL BACKGROUND	215
4.1.1	Basic Equations	215
4.1.2	Geometric Requirements	216
4.2	THE GEN1D INTERFACE	219
4.2.1	Setting Up Simulation Parameters	219
4.2.1.1	Simulation	219
4.2.1.2	Channel	223
4.2.1.2.1	Creating a Channel Object	223
• To create a new channel object:		223
4.2.1.2.2	Opening an Existing Channel Object	224
• To open an existing channel object:		224
4.2.1.2.3	Changing a Segment Attribute Value	224
• To change a segment attribute value:		224
4.2.1.2.4	Changing a Node Attribute Value	225
• To change a node's attribute values:		225
• To change a node's location or value:		225
4.2.1.3	Down Boundary	226
4.2.1.4	Up Boundary	227
4.2.1.5	Cross-Sections	228
4.2.1.5.1	Associating a Cross-Section with a Segment	228
• To associate a cross-section with a segment:		228
4.2.1.5.2	Scaling a Cross-Section	229
• To scale a cross-section:		229
4.2.1.5.3	Copying a Cross-section to a Segment	229
• To copy a cross-section to a segment:		229
4.2.1.5.4	Orthogonally Positioning a Cross-Section	230
• To orthogonally position a cross-section:		230
4.2.1.5.5	Resampling a Cross-Section	230
• To resample a cross-section:		230
4.2.1.5.6	Interpolating a Cross-Section	231
• To interpolate data from two cross-sections:		231
4.2.1.5.7	Vertically Offsetting a Cross-Section	232
• To vertically offset a cross-section:		232
4.2.1.5.8	Generating a Simple Cross-Section	232
• To generate a simple cross-section:		232
4.2.1.5.9	Removing a Cross-Section	233
• To remove a cross-section:		233
4.2.1.5.10	Cross-Section Properties	233
• To view the properties of a cross-section:		233
4.2.2	Running the GEN1D Model	234

• To run a GEN1D simulation:	234
4.2.3 Displaying Simulation Output	234
4.2.3.1 Creating a Hot Start From an Output	235
• To extract a Hot Start from a GEN1D model run:	235
5 THE HBV-EC MODEL	237
5.1 GENERAL BACKGROUND	237
5.1.1 Background and History of the Model	237
5.1.2 Algorithms Specific to the Model	237
5.1.2.1 Climate Zones	238
5.1.2.2 Snow Melt Factor Variation with Terrain Aspect and Slope	238
5.1.2.3 Watershed Routing	238
5.1.3 References	238
5.2 THE HBV-EC INTERFACE	238
5.2.1 The EnSim WaterShed Panel	239
• To identify an alternate basin object:	241
5.2.2 The Basin Panel	242
5.2.2.1 The Climate Tab	243
5.2.2.2 The Elevation Tab	243
5.2.2.3 The Land Use Tab	245
5.2.2.4 The Slope Tab	246
5.2.2.5 The Aspect Tab	247
5.2.2.6 Identifying Zones Within HBV-EC	248
• To identify a zone using polygons:	248
• To identify a zone using a GeoTIFF:	249
• To identify a climate zone using points:	250
5.2.3 The Simulation Panel	250
5.2.4 The Climate Zone Panel	253
5.2.4.1 The Parameters Tab	253
5.2.4.2 The Met Tab	257
5.3 THE HBV-EC MODEL	259
• To run the HBV-EC model:	259
5.3.1 The Results of the HBV-EC Model	260
APPENDIX A:FILE TYPES OF ENSIM CORE	263
Appendix A:General Information	263
• File Headers	264
• ASCII and Binary Files	268
• ASCII Files	268
• Binary Files	268
NATIVE FILE FORMATS	270
2D Rectangular Grids [r2s / r2v]	270
• File Headers [r2s / r2v]	270
• Data Organization [r2s / r2v]	271
Figure A.2:File Formats [r2s / r2v]	272

Figure A.2:ASCII	272
Figure A.2:Binary	272
• 2D Triangular Meshes [t3s / t3v]	274
• File Headers [t3s / t3v]	274
• File Formats [t3s / t3v]	275
• ASCII	275
• Binary	276
• Line Sets [i2s / i3s]	279
• File Headers [i2s / i3s]	279
• File Formats [i2s / i3s]	280
• ASCII	280
• Binary	280
• XYZ Point Sets [xyz]	281
• File Headers [xyz]	281
• File Format [xyz]	281
• XY Data Objects [xy] [dat]	282
• File Headers [xy] [dat]	282
• File Format [xy] [dat]	282
• Parcel Sets [pcl]	283
• File Headers [pcl]	283
• File Formats [pcl]	284
• ASCII	284
• Binary	284
• Point Sets [pt2]	286
• File Headers [pt2]	286
• File Formats [pt2]	287
• ASCII	287
• Time Series [ts1 / ts2 / ts3 / ts4 / ts5]	288
• File Headers [ts1 / ts2 / ts3 / ts4]	288
• File Headers [ts5]	289
• File Formats [ts1 / ts2 / ts3 / ts4 / ts5]	290
• ASCII	290
• Type 1 - *.ts1	290
• Type 2 - *.ts2	290
• Type 3 - *.ts3	291
• Type 4 - *.ts4	291
• Type 5 - *.ts5	292
• Binary	292
• Tables [tb0]	293
• File Headers [tb0]	293
• File Format [tb0]	294
• ASCII	294
• Binary	294
• Velocity Roses [vr1]	295
• File Headers [vr1]	295
• File Formats [vr1]	295

•	ASCII	295
•	Networks [n3s]	296
•	File Headers [n3s]	296
•	File Formats [n3s]	297
•	ASCII	297
•	Binary	298
•	2D Rectangular Cell Grids [r2c]	300
	Figure A.3:File Headers [r2c]	300
•	File Formats [r2c]	301
•	ASCII	301
•	Multiframe ASCII	302
•	Binary	302
•	SUPPORTED FOREIGN FILE FORMATS [ENSIM CORE]	304
•	GeoTIFF Theme files [thm]	305
APPENDIX B:FILE TYPES OF GREEN KENUE	307	
APPENDIX B:NATIVE FILE TYPES	308	
Appendix B:Watershed Objects [wsd]	308	
Appendix B:File Headers [wsd]	308	
•	File Format [wsd]	310
•	ASCII	310
Figure B.1:Binary	310	
FIGURE B.1:SUPPORTED FOREIGN FILE TYPES [GREEN KENUE]	311	
APPENDIX C:FILE TYPES OF THE RCA	313	
Appendix C:The Rating Curve Analysis File [rca]	313	
Appendix C:File Header [rca]	313	
•	File Format [rca]	314
•	ASCII	314
•	Binary	314
APPENDIX D:FILE TYPES OF GEN1D	315	
Appendix D:The GEN1D Parameter File	315	
Appendix D:File Header [g1d]	315	
Appendix D:File Format [g1d]	315	
Appendix D:Simulation Parameters	316	
Appendix D:General Parameters	316	
•	Simulation Parameters	316
•	Constants	317
•	Input Files	319
•	Boundaries	319
•	Output	321
APPENDIX E:FILE TYPES OF HBV-EC	323	
Appendix E:The HBV-EC Parameter Set File	323	
Appendix E:File Header [hbv]	323	
Appendix E:File Format [hbv]	326	

Table of Contents September 2010

Appendix E:The HBV-EC HBM File	327
Appendix E:File Header [hbm]	327
• File Format [hbm]	328
• The HBV-EC HBT File	329
• File Header [hbt]	329
• File Format [hbt]	330

1 ENSIM CORE

1.1 A QUICK OVERVIEW

1.1.1 The EnSim Simulation Environment

Developed at the Canadian Hydraulics Centre (CHC), EnSim was created to meet the needs of a wide range of environmental prediction and decision support systems. EnSim is designed as an advanced numerical modelling environment, as well as a general purpose data handling and visualisation system that can easily be adapted for any class of environmental data.

EnSim provides an ideal framework for the integration of environmental data, GIS information and model data. An EnSim application can be designed to run a simple numerical model or a suite of numerical models providing a variety of pre- and post-processing tools. It is designed as a generic toolkit from which a system developer chooses components to create an application.

For the modeller, EnSim creates a virtual environment where simulation results can be viewed, animated and analyzed in one, two and three dimensions. This allows you to observe complex interactions of various phenomena in an intuitive manner, providing a realistic view of simulation results.

Presentation of simulation results to non-technical audiences can be greatly improved by providing seamless integration with other Windows applications such as word-processors, spreadsheets and multimedia tools.

1.1.2 Getting Started

EnSim Core forms the basis of a variety of applications (e.g. WaveSim, Blue Kenu, Green Kenu) that comprise the EnSim family. These applications all share fundamental functions, which form the core of EnSim. As a result, this manual is set up in a modular fashion. There is a section under the heading "EnSim Core" that details the functions common to all EnSim applications, and a separate section, under the title of the application, that describes those functions that are particular to the specific application.

If you're a first time user, it might be easier to begin with the section on EnSim Core to become familiar with the basics of EnSim before proceeding to the section specific to the EnSim application. The sections of this manual that are specific to a particular application illustrate how to perform only the functions that are specific to that application (e.g. Blue Kenu) and assume that you are familiar with core EnSim functions.

1.1.3 Getting Help with EnSim

EnSim is a Windows-based application. All EnSim documentation assumes that you are familiar with Windows-based applications. That is, it assumes you know how to use a mouse, open a menu, choose menu and dialog options, and other Windows-based functions. EnSim documentation also assumes familiarity with standard Windows menus and buttons, such as the **Open** document command, which can be found in the **File** menu or by clicking the  button. Consult your Windows documentation for help in using Windows-based applications.

EnSim documentation consists of a manual and an online help system. The online help system is accessed through the EnSim **Help** menu. The manual and the online help system of EnSim documentation are intended to be independent. All information contained in the manual can also be found in the online help system. Version and copyright information about EnSim can be obtained using the command **About...** in the **Help** menu.

1.1.3.1 Conventions in EnSim Help

Bold designates the name of a menu, menu choice, dialog, dialog option, or workspace category.

e.g. **File**

Italics highlight a term or concept being defined or described.

e.g. *Categories* are elements defined by EnSim to organise the workspace.

<Angle brackets> indicate key presses.

e.g. <Ctrl> key

An “→“ indicates a sequence of menu selections.

e.g. **File**→**Open**

1.2 THE WORKSPACE

The *WorkSpace* provides an organizational structure for the data files and view windows being used in EnSim. In a window on the left-hand side of the screen, the WorkSpace is displayed as a tree consisting of a hierarchical display of categories (organizational headings for objects) and objects (data or view objects), similar to the file hierarchy structure of Windows Explorer. Unlike Windows Explorer, the hierarchy does not represent the physical location of files; rather, it represents the relationship between the objects and between objects and views. The top of the tree is always the WorkSpace, represented by the  icon. The workspace can be toggled on or off using the **WorkSpace** command in the **View** menu.

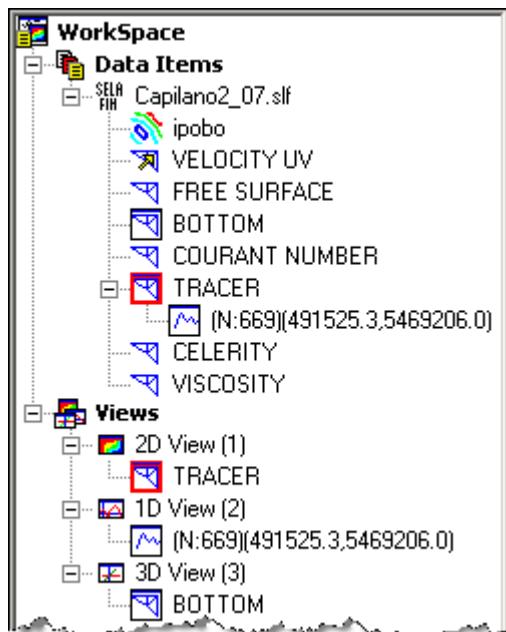


Figure 1.1: The EnSim WorkSpace includes Data Items and Views

The second branch of the hierarchical tree consists of *categories*, which are organizational headings for objects. There are two categories in the EnSim WorkSpace. **Data Items**, represented by the  icon, is the first category. It contains all the files, or *data items*, that have been opened or created during the EnSim session. The second category is **Views**, represented by the  icon. It contains all the view windows that are active during the session and the objects associated with each view.

The remaining branches of the hierarchical tree are *objects*. Objects can represent files, parts of a file, or views. An example of an object would be a model results file in the **Data Items** category. Another example would be a view window in the **Views** category.

1.2.1 Managing Objects in the WorkSpace

Objects sometimes contain other objects. An example of an object containing another object is a Green Kenue watershed object, shown below.



Figure 1.2: A Green Kenu watershed object contains other objects

The three objects under the Test Watershed object are considered children of the parent object. Other examples of objects that are shown as children are extracted time series and 3D line sets. The children, or components, of an object can be displayed or hidden in the workspace by clicking on the + or - signs, respectively, located to the left of the object. In many cases, only the children of an object can be dragged into a view. If a child was created from a viewable parent object, such as a time series extraction from a triangular mesh, then both the child and parent can be displayed.

Objects in the WorkSpace are represented by icons, which indicate the object's type, and therefore, some of the object's properties. Icons for some common data items are detailed below. Details concerning each type of data file can be found in the Appendices.

- █ a file, usually a container file for other objects
- █ rectangular grid, scalar data
- █ rectangular grid, vector data
- █ triangular mesh, scalar data
- █ triangular mesh, vector data
- █ 2-dimensional line set (for example, isolines or GIS data)
- █ 3-dimensional line set
- █ network file (describes a network of segments and nodes)
- █ xy data item
- █ time series, scalar
- █ time series, vector
- █ point set
- █ geoTIFF image
- █ table

There are a few icon decorations that indicate the status of an object:

- a black square █ indicates that a data item is in a view window.
- a red square █ indicates that animation is activated on an object in a view window.
- a red circle █ indicates an empty object, which contains no data

- a yellow star  in the bottom left-hand corner of an icon indicates that the data item is in the process of being created. For example, a new regular grid will have a yellow star, while a regular grid loaded into EnSim with the **Open** file command will not have a yellow star.

When an object is selected in the WorkSpace, it is highlighted and becomes the current object. All functions are then applied to that object.

Objects in the WorkSpace can be manipulated in the following ways:

- **Adding an object to another object.** Data items can be added to the **Data Items** hierarchy by opening a file or creating a new data item, such as a grid. Files can be opened in the same manner.

New files can be created by choosing **File**→**New** and selecting the item to be created, or by creating the data item with an EnSim function. For example, when a 3D line is created, it is added to the WorkSpace as a child of the parent object. Selected objects can be added to other objects in the WorkSpace by dragging and dropping.

An object can only be dragged into another object that is capable of receiving data, such as a view window or an empty data item.

- **Adding data items to a view.** To add an object to a view, select it from the **Data Items** section of the WorkSpace, drag it to the **View** section of the WorkSpace and drop it into a view object. The default view object in EnSim is a 2D View window.

After an object has been dropped into a view, it can be displayed or hidden without removing it from the view by toggling the **Visible** command in the shortcut menu, the **Edit** menu, or the **Display** tab of the object's Properties dialog box.

A data item can only be displayed in one view window at a time. To display the data item in multiple windows, a copy of the file must be opened for each view window. For example, if a rectangular grid is to be displayed in three view windows, there must be three copies of the rectangular grid displayed in the WorkSpace under **Data Items**. Each copy of the grid is then dropped into one of the appropriate view windows.

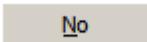
- **Removing an object from another.** Data items can be removed from the **Data Items** hierarchy by selecting the data item in the WorkSpace and using the <Delete> key or the **Remove** command in the shortcut menu or the **Edit** menu. Removing a data item from **Data Items** has the effect of removing or closing the object from the application.
- **Removing data items from a view.** Data items can be removed from a view by selecting the data item within the list of views and using the <Delete> key, by selecting **Remove** in the data item's shortcut menu, or by selecting **Edit**→**Remove** from the menu bar.
- **Viewing an Object's properties.** Double clicking on an object opens its Properties dialog. The Properties dialog of a selected object can also be accessed from the **Edit** menu or the object's shortcut menu.
- **Viewing an object's shortcut menu.** Right-clicking on an object displays its shortcut menu.

- **Renaming an object.** After opening the Properties dialog, select the **Meta Data** tab and change the text in the **Title** or **Name** fields. The **Title** field will change the name of the object at the top of the Properties dialog box, while the **Name** field will change the name of the object in the WorkSpace, data probes, and the automatic title of the colour-scale legend.

1.2.2 Saving and Loading The WorkSpace

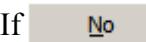
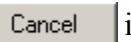
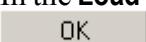
EnSim allows you to save the current state of the WorkSpace to an EnSim WorkSpace File (*.ews). The EnSim WorkSpace File contains all the data item settings, including colour scales, legend options, scaling, rendering options, line width, point size, and so on, for all currently opened objects. As well as saving data item settings, it also saves the settings for all the views and the object view relationships. This ASCII file should not be edited directly.

To Save a WorkSpace:

1. On the menu bar, select **File→Save WorkSpace...**
2. When the **Save Current WorkSpace As** dialog appears, enter an appropriate name and click .
3. The WorkSpace can only be saved if all objects have a file association. Extracted isolines, time series, new Point Sets, new Line Sets, etc. do not have such an association when first created. If any of the objects within the WorkSpace need to be saved, you will be prompted to do so. Click  to save the objects, or  to cancel the Save WorkSpace operation.
1. Give each of the objects an appropriate name and click  to save them. If you click  for any object, the Save WorkSpace operation will be cancelled, but any objects already saved will remain saved.

To Load a WorkSpace:

Note: Loading an EnSim Workspace file will remove any existing objects or views from the EnSim environment. If necessary, make sure that you've saved the current WorkSpace. EnSim WorkSpace files created by a EnSim application can be loaded by that EnSim application only. The files are not compatible in any other EnSim application.

1. Select **File→Load WorkSpace** from the menu bar.
2. When prompted to continue, select  or . The current WorkSpace will be cleared.
If  or  is selected, no changes will be made to the current WorkSpace.
3. In the **Load WorkSpace From** dialog, select the desired EnSim WorkSpace File and select the .

1.3 THE ENSIM INTERFACE

The interface is customized for specific applications as necessary. The basic graphical user interface consists of four main components: a menu bar and a tool bar, both for selecting various windows and EnSim functions; a workspace, for managing open data files and views; and an area for various views (i.e., 1D, Polar, 2D, 3D, Spherical, and Report).

An example of the basic EnSim interface follows. The title bar identifies this interface as being from WaveSim. However, most EnSim applications have the same basic interface and will look similar to this example.

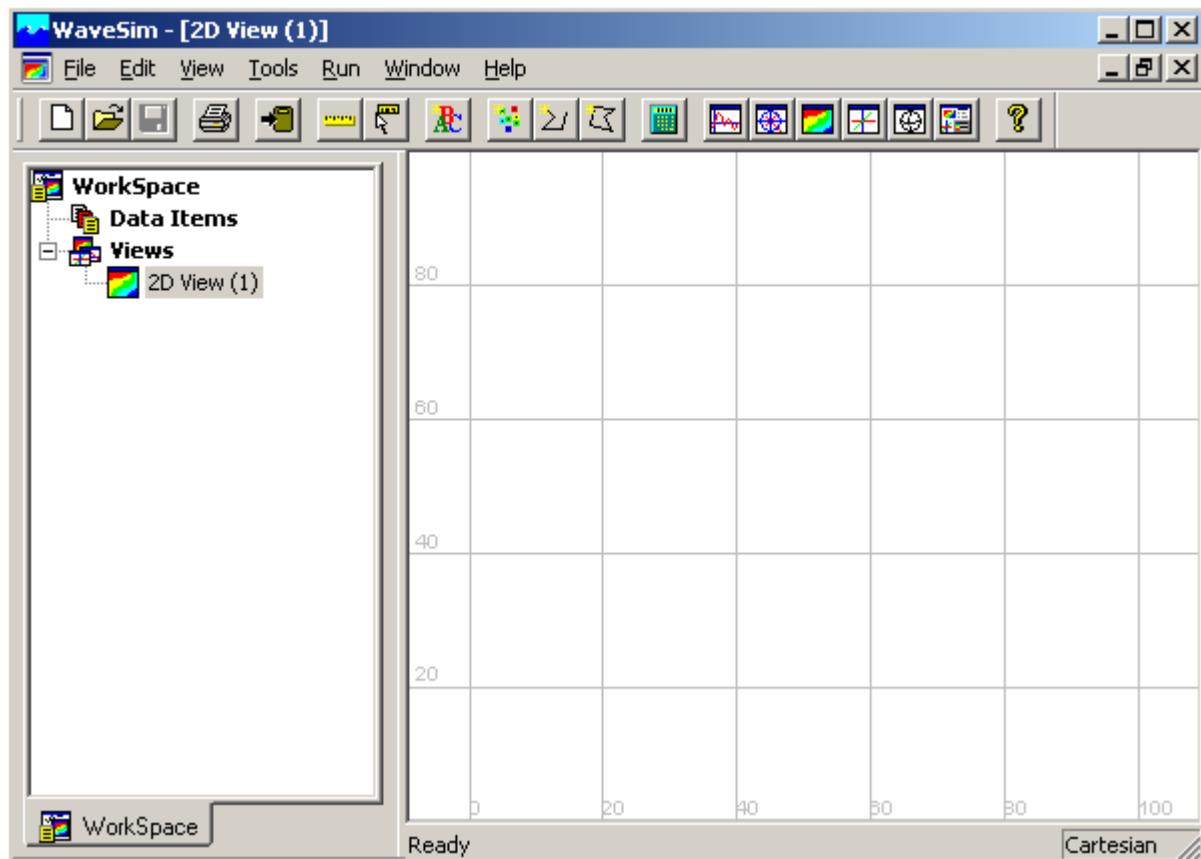
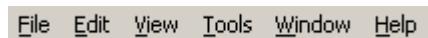


Figure 1.3: The EnSim interface window

1.3.1 The Menu Bar

The *menu bar* consists of the standard Windows options: **File**, **Edit**, **View**, **Tools**, **Window**, and **Help**.



Commands in these menus that are specific to EnSim will be detailed in the appropriate sections. Specific EnSim applications may contain other menus in the menu bar (e.g. WaveSim

and OilSim have the **Run** option in their Menu Bars, which contains commands related to running a simulation).

1.3.2 The Tool Bar

The main *tool bar* gives quick access to some of the commands in the menus. It can be toggled on or off using the **Tool Bar** command in the **View** menu. To move the tool bar, click on the tool bar with the mouse and drag it to the desired location.



Other tool bars exist for specific command functions. For example, there is an Animation tool bar for EnSim applications that have animation capabilities. For more information on these tool bars, see the information specific to the function.

1.3.3 The T3 Mesh Editing Toolbar

The T3 Mesh Toolbar makes available several mesh editing functions. It can be turned on or off by selecting **View→T3 Mesh Toolbar** from the menu bar. Like the main Tool Bar, it can be moved around by clicking and dragging on its handle. For more information on the T3 Mesh Toolbar, see "Editing T3 Meshes", on p. 79.



1.3.4 Shortcut Menus

Shortcut or *context menus* are available for most windows and objects by right clicking on the selected window or object.

1.4 DATA ITEMS

The items listed under the category **Data Items** in the WorkSpace are referred to as objects. The following is an image of a Green Kenu WorkSpace displaying various objects (i.e. the 'Jock River' watershed object, the 'Rectangular Grid', the 'Basin 8 boundary', etc.).

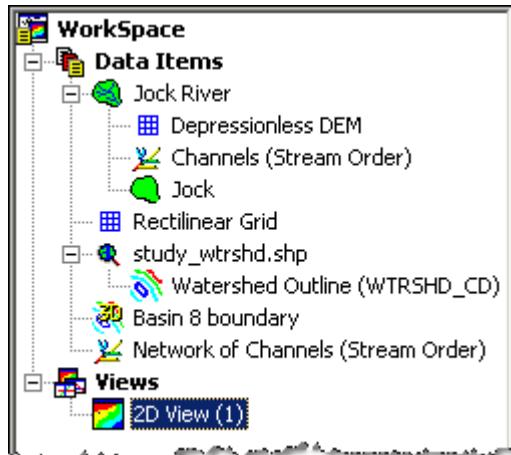


Figure 1.4: The Green Kenu WorkSpace contains several types of Data Items

Each object is a coherent collection of data. The data contained within a particular type of object may come from a variety of sources. Take a 2D line set object, for example. A 2D line set object consists of one or more 2-dimensional lines. The geometry of each line is defined by two or more xy points. Each line may also have a number of attributes associated with it. For example, if the line set is a set of isolines representing contour data, each line will have an associated elevation. The data that comprises a line set object may come from, for instance, an ArcInfo shape file, a MapInfo interchange file, or an EnSim native i2s (2D line set) file. The organization of data is quite different in each of these source files. However, the data from each are organized in EnSim as a line set object. EnSim uses objects as a way of taking data in various formats, and putting them in a uniform format.

All objects of the same type (e.g. line sets, 2D Rectangular scalar grids, point sets, etc.) can be used and displayed in the same way and they can all have the same functions applied to them. For example, all line set objects have the same options for display, and can be used in performing the same functions. The same display options and functions are not necessarily applicable to an object of a different type, say a 2D grid object or a point set.

All EnSim objects have a native file format. See "Native File Formats", on p. 270, for details.

There are different categories of objects: spatial objects and container objects. Spatial objects are those that have geometry and attributes. They are the ones that can be displayed in a view, edited, manipulated, etc. Container objects do not have geometry. They are containers or organizers for other objects and data. They keep related objects together in one location. A Green Kenu Watershed object is a container, and so is a Blue Kenu SELAFIN (*.slf) object. Time series are similar to spatial objects in that they may be displayed, edited, and manipulated, but they are different in that they do not have geometry, only attributes.

1.4.1 Loading and Importing Data Items

To load a data item into the WorkSpace, there are two types of data items recognized by EnSim:

1. For native EnSim data items, choose the **Open** command from the **File** menu, or use the  button. When the Open dialog box appears, select the file to be opened into the WorkSpace and choose the  button. The 8 files most recently opened in EnSim are shown at the bottom of the **File** menu.
2. For foreign data items, choose the **Import** command from the File menu. When the Open dialog appears, select the file and choose the  button.

1.4.1.1 Native Data Items

The data items that are recognized by all EnSim applications are:

2D Rectangular Scalar Grids: (r2s) Two-dimensional rectangular or regular grid having evenly spaced nodes in both dimensions. X-spacing may differ from y-spacing. The node values of the grid are scalar quantities (e.g. Elevation, concentration etc.) associated with each node. May be time-varying. Represents a continuous surface.

2D Rectangular Vector Grid: (r2v) Two-dimensional rectangular or regular grid having evenly spaced nodes in both dimensions. X-spacing may differ from y-spacing. The node values of the grid are vector components in the x and y direction (e.g. velocity, flux, etc.). May be time-varying.

2D Rectangular Cell Grids: (r2c) Two-dimensional rectangular or regular grid having evenly spaced nodes in both dimensions. X-spacing may differ from y-spacing. The node values of the grid are scalar quantities (e.g. Land-use, etc.) and are constant over each cell. May be time-varying. Represents a discrete surface.

2D Triangular Scalar Mesh: (t3s) Two-dimensional triangular mesh. The node values of the mesh are scalar quantities (e.g. elevation, concentration etc.) associated with each node. May be time-varying. Represents a continuous surface.

2D Triangular Vector Mesh: (t3v) Two-dimensional triangular mesh. The node values of the mesh are vector components in the x and y direction (e.g. velocity, flux, etc.). May be time-varying.

2D Line Sets: (i2s) Open or closed collection of lines defined by two-dimensional nodes. Each line may have multiple associated attributes. For example, if the lines are contour lines it may include elevation data. Attributes may be integer, float, text, etc. See "Line Sets [i2s / i3s]", on p. 279, for further details.

3D Line Sets: (i3s) Open or closed collection of lines defined by three-dimensional nodes. Each line may have multiple associated attributes. For example, if the lines are contour lines it may include elevation data. Attributes may be integer, float, text, etc. See "Line Sets [i2s / i3s]", on p. 279, for further details.

Point Sets: (pt2) Set of points, each represented by an x and y coordinate. Each point may have multiple associated attributes.

XYZ Point Sets: (xyz) Set of points, each represented by an x, y, and z coordinate.

Parcel Sets: (pcl) Set of points, each represented by an x, y, and z coordinate. May have multiple attributes, and may be time-varying. Location of points may move in time.

XY Data Sets: (xy) Set of scalar pairs. Each pair represents values from two attributes, attribute X and attribute Y.

Scalar Implicit Time Series: (ts1) Represents a scalar quantity varying with a constant time step.

Vector Implicit Time Series: (ts2) Represents a vector quantity varying with a constant time step.

Scalar Explicit Time Series: (ts3) Represents a scalar quantity with a varying time step.

Vector Explicit Time Series: (ts4) Represents a vector quantity with a varying time step.

Networks: (n3s) A set of connected Segments (polylines) connected at Nodes. Each Segment is made up of a series of 3D vertices, and may have multiple attributes (e.g. roads, channels). Nodes may have multiple attributes.

Tables: (tb0) A set of data values organized into rows and columns. Columns represent the attributes and rows represent the values at each attribute index.

Velocity Roses: (vr1) Represents probabilities of vector quantities tabulated by magnitude and direction.

1.4.1.2 Foreign Data Items

Please refer to "Supported Foreign File Formats [EnSim Core]", on p. 304, for further details.

1.4.2 Saving and Exporting Data Items

To save a data item to a file, select the object in the WorkSpace. To save the current object, choose the **Save** command from the **File** menu or use the  button. A copy of the object may be saved with the **Save Copy As...** command from the **File** menu. When the **Save Copy As...** command is used, a copy of the current object is saved. This command is used to save a back-up copy of an object and then to continue to edit the original object, or to export the object to another file format.

All data items, regardless of their source, can be saved in at least one of the native EnSim file formats. The format in which the object may be saved depends on the type of data. Click on the  button or choose **Save** or **Save Copy As...** option from the **File** menu. Use the **Save as type** box at the bottom of the dialog window to view the various file formats in which the object may be saved. See Appendix A for a complete description of the native file formats.

The data items and the file formats in which they may be saved are as follows:

Icon	Object Type	May Be Saved As...
	Point Set	.pt2 - ASCII (EnSim format)
		.xyz - ASCII (EnSim format)
		.shp - ArcView Shape Format
		.mif - MapInfo Interchange Format
	XYZ Point Set	.xyz - ASCII (EnSim format)
		.shp - ArcView Shape Format
		.mif - MapInfo Interchange Format
	Parcel Set	.pcl - Lagrangian parcel set (EnSim format)
		.shp - ArcView Shape Format
		.mif - MapInfo Interchange Format
		.mif - Multi-frame MapInfo Format
	XY Data Items	.xy or .dat ASCII (EnSim format)
	2D Rectangular Scalar Grid	.r2s - ASCII Single Frame (EnSim format)
		.r2s - Binary Single Frame (EnSim format)
		.r2s - Binary Multi Frame (EnSim format)
		.t3s - ASCII Single Frame (EnSim format)
		.t3s - Binary Multi Frame (EnSim format)
		.xyz - ASCII (EnSim format)
		.grd - Surfer Grid Format
		.asc - ArcInfo ASCII Grid Format
		.r2c - Rect2D Cell ASCII Single Frame
		.r2c - Rect2D Cell ASCII Multi Frame
		.r2c - Rect2D Cell BINARY Single Frame
		.r2c - Rect2D Cell BINARY Multi Frame
		.tif - GeoTIFF Format (if data range 0-255)

Icon	Object Type	May Be Saved As...
	2D Rectangular Vector Grid	.r2v - ASCII Single Frame (EnSim format)
		.r2v - Binary Single Frame (EnSim format)
		.r2v - Binary Multi Frame (EnSim format)
		.r2s - ASCII Magnitude (EnSim format)
		.r2s - Binary Magnitude (EnSim format)
		.t3v - ASCII Single Frame (EnSim format)
		.t3v - Binary Multi Frame (EnSim format)
	2D Triangular Scalar Mesh	.t3s - ASCII (EnSim format)
		.t3s - Binary Single Frame (EnSim format)
		.t3s - Binary Multi Frame (EnSim format)
		.xyz - Magnitude (EnSim format)
		.ngh - TriGrid Neigh Format
		.nod - TriGrid Node Format
		.kml - Google Earth Format
	2D Triangular Vector Mesh	.t3v - ASCII (EnSim format)
		.t3v - Binary Single Frame (EnSim format)
		.t3v - Binary Multi Frame (EnSim format)
		.t3s - ASCII magnitude only (EnSim format)
		.t3s - Binary magnitude only (EnSim format)
		.xyz - ASCII magnitude only (EnSim format)
	Network	.n3s - ASCII (EnSim format)
		.n3s - Binary Single Frame (EnSim format)
		.n3s - Binary Multi Frame (EnSim format)
		.i2s/.i3s - ASCII (EnSim format)
		.xyz ASCII (EnSim format)
		.shp - ArcView Shape Format
		.mif - MapInfo Interchange Format
		.kml - Google Earth Format

Icon	Object Type	May Be Saved As...
	Time Series Type 1	.ts1 - ASCII (EnSim format)
		.ts3 - ASCII (EnSim format)
	Time Series Type 3	.ts3 - ASCII (EnSim format)
	Time Series Type 2	.t2s - ASCII Mag. and Dir. (EnSim format)
		.t2s - ASCII U and V (EnSim format)
		.t4s - ASCII Mag. and Dir. (EnSim format)
		.t4s - ASCII U and V (EnSim format)
	Time Series Type 4	.t4s - ASCII Mag. and Dir. (EnSim format)
		.t4s - ASCII U and V (EnSim format)
	2D LineSets	.i2s - ASCII (EnSim format)
		.i3s - ASCII (EnSim format)
		.xyz - ASCII (EnSim format)
		.shp - ArcView Shape Format
		.mif - MapInfo Interchange Format
		.kml - Google Earth Format
	3D LineSets	.i3s - ASCII (EnSim format)
		.i2s - ASCII (EnSim format)
		.xyz - ASCII (EnSim format)
		.xy - ASCII distance value (EnSim format)
		.shp - ArcView Shape Format
		.mif - MapInfo Interchange Format
		.kml - Google Earth Format
	Tables	.tb0 - Table Data (EnSim format)
		.csv - Comma Delimited text
	Velocity Rose	.vr1 - ASCII (EnSim format)
	ArcView Shape Files	None *

Icon	Object Type	May Be Saved As...
	MapInfo Interchange Format	None †
	GeoTIFF	tiff - GeoTIFF ‡
	Surfer Grid	.r2s - ASCII Single Frame (EnSim format)
		.r2s - Binary Single Frame (EnSim format)
		.r2s - Binary Multi Frame (EnSim format)
		.t3s - ASCII Single Frame (EnSim format)
		.t3s - Binary Multi Frame (EnSim format)
		.xyz - ASCII (EnSim format)
		.grd - Surfer Grid
		.asc - ArcInfo ASCII Grid
	DEM (DTED & CDED)	.r2s - ASCII Single Frame (EnSim format)
		.r2s - Binary Single Frame (EnSim format)
		.r2s - Binary Multi Frame (EnSim format)
		.t3s - ASCII Single Frame (EnSim format)
		.t3s - Binary Multi Frame (EnSim format)
		.xyz - ASCII (EnSim format)
		.grd - Surfer Grid
		.asc - ArcInfo ASCII Grid
	ArcInfo ASCII Grid	.r2s - ASCII Single Frame (EnSim format)
		.r2s - Binary Single Frame (EnSim format)
		.r2s - Binary Multi Frame (EnSim format)
		.t3s - ASCII Single Frame (EnSim format)
		.t3s - Binary Multi Frame (EnSim format)
		.xyz - ASCII (EnSim format)
		.grd - Surfer Grid
		.asc - ArcInfo ASCII Grid
	Binary Raster	.r2s - ASCII Single Frame (EnSim format)

Icon	Object Type	May Be Saved As...
		.r2s - Binary Single Frame (EnSim format)
		.r2s - Binary Multi Frame (EnSim format)
		.t3s - ASCII Single Frame (EnSim format)
		.t3s - Binary Multi Frame (EnSim format)
		.xyz - ASCII (EnSim format)
		.grd - Surfer Grid
		.asc - ArcInfo ASCII Grid
■■	SRTM1 & SRTM3	.r2s - ASCII Single Frame (EnSim format)
		.r2s - Binary Single Frame (EnSim format)
		.r2s - Binary Multi Frame (EnSim format)
		.t3s - ASCII Single Frame (EnSim format)
		.t3s - Binary Multi Frame (EnSim format)
		.xyz - ASCII (EnSim format)
		.grd - Surfer Grid
		.asc - ArcInfo ASCII Grid

Note:

* ArcView Shape File objects cannot be saved. The child objects can be saved if they are of type .xyz, .i2s, or .i3s. ArcView Shape File objects can contain multiple objects, but all these children must be the same file type.

† MapInfo Interchange Format objects cannot be saved. The child objects can be saved if they are of type .xyz, .i2s, or .i3s. MapInfo Interchange Format objects can contain multiple objects, and these children can be different file types.

‡ GeoTIFFS, although not considered a native EnSim file format, can be saved within EnSim. Any changes made to the GeoTIFF's legend or colours will be preserved within the file. Opening a GeoTIFF that has been saved in EnSim with another application may remove this data.

Hint: To save the contour lines of a gridded data item, refer to the section on "Extracting Isolines" under Extracting Data, on p. 95. Extract the isolines and save them as a line set (.i2s, .i3s).

1.4.3 Properties of Data Items

All objects have properties that can be viewed and edited from the Properties dialog box. An object's Properties dialog box can be accessed in three ways:

- With the object selected, choose the Properties command from the **Edit** menu. An object is selected using the workspace, or by double clicking on the object in a view window.
- With the object selected, choose the **Properties** command from the shortcut menu (right click). An object is selected using the workspace, or by double clicking on the object in a view window.
- Double click on the object in the workspace.

The tabs contained in a properties dialog are specific to the object's data type (e.g. point data, line set, triangular mesh). Objects that can be displayed in 1D usually have three tabs: Display, Data, and Meta Data. Objects that can be displayed in a 2D or 3D view window commonly have five tabs: Display, Colour Scale, Data, Spatial, and Meta Data.

Some objects will have none of the tabs described below. For example, some objects that are specific to one application are designed to guide you through a specific set of tasks. These details are described in the section of the manual that is specific to the particular application. Some Properties tabs may be quite different for some objects, especially those that are specific to one EnSim application.

The Properties tabs may also be embedded in dialogs that are specific to a particular application. For example, a Parameter Set dialog has the "Standards Tabs" along with other meta data such as variable names and values.

1.4.3.1 Display Properties

The display tab allows you to change the rendering style of the object, as well as the vertical position. Vertical position only applies to objects being displayed in a 3D window. The display tab also allows you to change the display style of lines and/or points.

Display options are similar for objects that have similar characteristics. For example, an ArcView shape file and an EnSim isoline will have similar display properties because both are line sets. Recall that similar objects have the same icon displayed in the WorkSpace.

The following figure is an example of a **Display** properties tab:

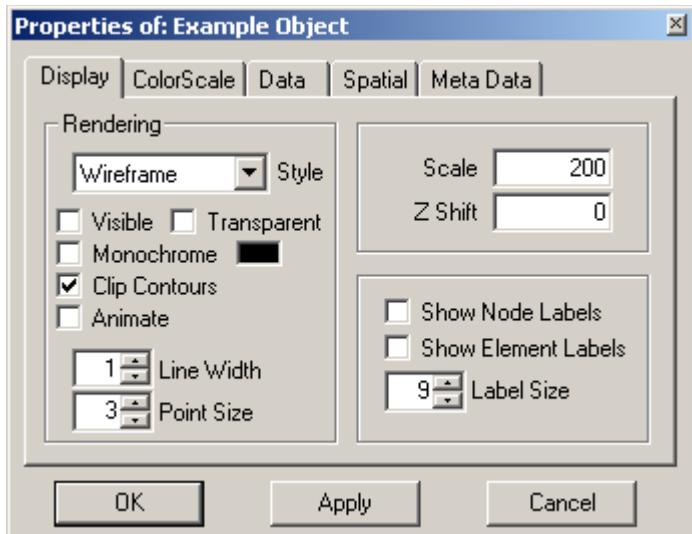


Figure 1.5: The Display tab of the Properties dialog box

1.4.3.1.1 Rendering Options

- **Style:** Changes the way a data item is represented. There are many different display styles. The availability of styles in the pull-down menu depends on the type of object. Not all styles are available for a data item type.

These style types may be applied to objects whose icon represents **grid data**, such as a regular grid or triangular mesh. These objects have the following icons: .

- **Wireframe** - Shows the lines comprising the grid without shading.
- **Surface** - Shaded surface representation of a grid object.
- **Isolines** - Isolines are line contours in which the levels are determined by the object's colour scale.
- **Filled contours** - Filled contours are isolines or contours with the areas between the lines filled in with the colour of the contour level. As with isolines, filled contour levels are determined by the object's colour scale.
- **Points** - Shows the nodes of a grid.
- **Arrows** - Only applicable to vector objects, arrows represent the vector at each node. The size and colour of the arrow represent the magnitude of the vector.

These style types may be applied to objects whose icon represents a **line set** or **time series**. Icons for these objects include: .

- **Lines** - Shows only the lines or polylines of the object. Points are hidden.
- **Lines and Points** - Shows the lines or polylines and points of the object.
- **Points Only** - Shows only the points or nodes of the lines or polylines. Lines are hidden.

These style types may be applied to objects whose icon represents **point data**, , **line sets**, , and **time series**, , displayed as lines and points, or points only. For the line sets and time series, these styles are available in the lower right portion of the window.

- **Line Styles:** Solid, Dotted, Short Dash, Long Dash, Dot Dash, and Long Dot Dash
- **Point Styles:** Fill Square, Half Fill Square, Square, Fill Triangle, Half Fill Triangle, Triangle, Fill Marker, Half Fill Marker, Marker, Fill Diamond, Half Fill Diamond, Diamond, Fill Octagon, Half Fill Octagon, Octagon, Fill Star, Star, X, Plus, and Asterisk
- **Visible:** Indicates whether the object that has been dragged into a view is visible in a new window. This option can also be selected from the shortcut menu.
- **Transparent:** Indicates whether the object is transparent to underlying objects in the view window.
- **Monochrome:** Changes the colour display of the object to one colour. The colour can be changed by clicking on the adjacent colour box.
- **Clip Contours:** Only available for grid objects. Only available when the display style is "Filled Contours." When this is toggled on, areas lower than the lowest contour level are not displayed.
- **Animate:** Toggles the animation of time-varying data. The object is not in animation mode by default.
- **Line Width:** Adjusts the line size of lines in the image. Maximum line width is 10.
- **Point Size:** Adjusts the point size of points in the image. Maximum point size is 20.

1.4.3.1.2 Vertical Display Options

- **Scale:** Controls the exaggeration of magnitude. A small number reduces the exaggeration, while a large number will increase it.
 - **Scalar Data:** In a 3D view, the magnitude refers to the value of the z-coordinate or data attribute, such as elevation or concentration.
 - **Vector Data:** In either 2D or 3D, the magnitude refers to the length of the vector arrows.
- **Shift:** Changes the vertical location of the object in a 3D view. The default shift is always zero. A positive shift moves the object in the positive z-axis direction, and a negative shift moves the object in the negative z-axis direction.

1.4.3.1.3 Other Display Options

- **Grid Step:** This is an option only for regular grids. The default display is 1. This means that every node of the grid is displayed. If the grid file is very large, EnSim may be slow in updating the view when it contains the grid. The grid step can be increased to show fewer points and allow EnSim to update the display more quickly. The usefulness of this feature is dependent on your machine's CPU and graphical capabilities.
- **Points:** This option is available to line sets. When the **Lines and Points** or **Points** style is chosen under the **Rendering** option, this option allows the display of the points to be

changed. For example, the points can be displayed as an X, as triangles, as squares, and so on.

- **Show Node Labels:** This is an option for gridded objects. If this option is selected, the node i.d. number is displayed at each node. The size of the labels can be adjusted.
- **Show Element Labels:** This is an option for gridded objects. If this option is selected, the node i.d. number is displayed in the centre of each element. The size of the labels can be adjusted.

1.4.3.2 Colour Scale

A *colour scale* consists of a colour spectrum and colour levels. It is automatically generated upon opening or creating a data item, and can be edited. The colour scale legend can be created from the colour scale and displayed in a view window. See the section "Legends" under View Decorations, on p. 54, for more information.

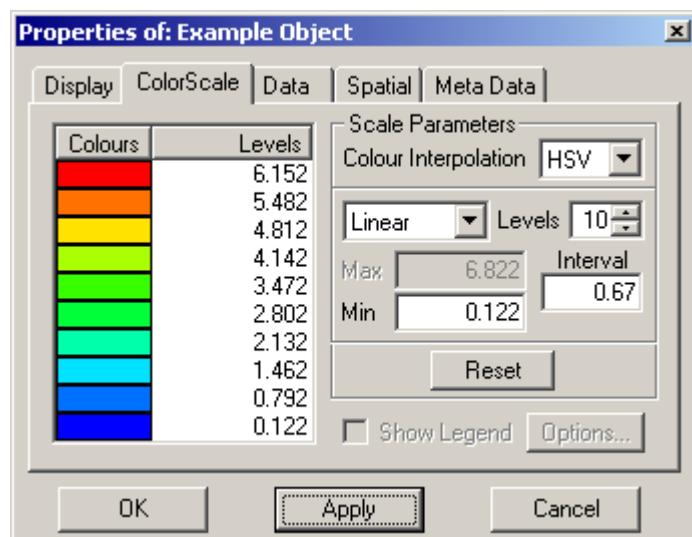


Figure 1.6: The Colour Scale tab of the Properties dialog box

To edit the colour scale:

Open the Properties dialog box for the object. Select the **Colour Scale** tab.

The following fields may be edited:

- **Colours:** This is the colour spectrum. Clicking on an individual colour box provides a dialog to modify its colour. Changing the top or bottom colour, and then clicking on the **Colours** heading will modify the entire spectrum. This will cause the colours to be interpolated between the top and bottom colours.
- **Levels:** This is the listing of colour levels. Clicking on an individual level will allow you to edit its value. Clicking on the **Levels** heading interpolates the levels based on the Scale Style, Levels, Max., Min., and Interval.
- **Scale Parameters:** These parameters are used to define the colour levels.

- **Colour Interpolation:** Determines the type of colour spectrum interpolation
 - **RGB:** Linear Red, Green, and Blue interpolation.
 - **HSV:** Linear Hue, Saturation, and Value interpolation.
- **Style:** Determines the type of level interval.
 - **Linear** sets a linear scale. This style is the default.
 - **Nlog** sets a natural logarithmic scale.
 - **Quadratic** sets a quadratic scale.
- **Levels:** Determines the number of levels. The maximum number of levels available is 40. When more than 10 levels are used, the additional levels will initially appear black. They should be customized before being used.
- **Max.:** Determines the value of the highest level. When the style is Linear, this value is shaded and cannot be edited. It is calculated automatically from the Min. and Interval parameters.
- **Min.:** Determines the value of the lowest level.
- **Interval:** Determines the interval value between levels. When Nlog or Quadratic styles are applied, this value is shaded, and cannot be edited. Under those conditions, it will be calculated based on the Max. and Min. parameters.
- **Reset:** This button will return all values in the dialog to their original defaults.
- **Show Legend:** Shows the object's colour scale as a legend in the current view window. For more information, see the section "Legends" under View Decorations, on p. 54.
- **Options:** Opens a dialog box for colour scale legend options.

To apply a previously created colour scale:

See the section "Copying Data Item Properties" under Properties of Data Items, on p. 31.

1.4.3.3 Data Attributes

The **Data** tab provides information on the data contained in the object. A data item may have multiple attributes. A typical data tab is shown below.

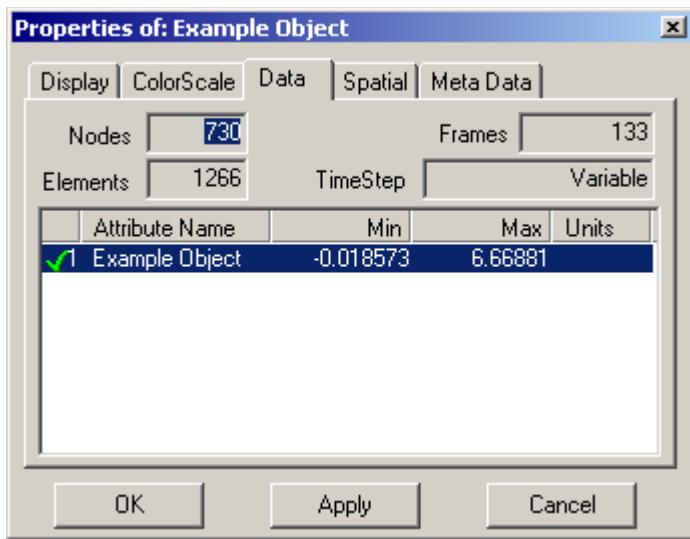


Figure 1.7: The Data tab of the Properties dialog box

The various data attributes associated with the object are listed under **Attribute Name**. Attributes may be numeric values such as elevations, or text such as the name of a river. If the attribute is a number, the minimum and maximum values are displayed. If the attribute is text, zero is entered under the **Min.** and **Max.** headings. If there are units associated with an attribute and if they are appropriately specified in the data file they are displayed under **Units**.

The *current attribute* is the active attribute of an object. It is the data that is displayed, edited or modified when using EnSim's tools. Each data attribute has its own colour scale. When the colour scale is edited, it modifies the colour scheme for the current attribute only. When editing an object, only the data of the current attribute is modified. Tools used on a selected object, like the Map Object or Extract Time Series tools, only modify or use the current data attribute of the object.

If an object has multiple attributes, the name of the current data attribute is shown in (parentheses), to the right of the object name in the WorkSpace, as shown below.

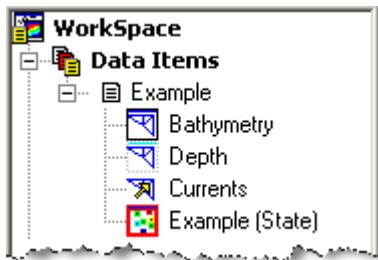


Figure 1.8: The current attribute of the "Example" object is State

In the figure above, the Point Set object entitled "Example" is the only object in the WorkSpace possessing multiple attributes. The current data attribute, State, is shown beside the object's name in the WorkSpace.

To change the current attribute, open the object's **Properties** dialog and select the **Data** tab. Choose the desired attribute and click the **Apply** button. Alternatively you can double

click on the attribute name in the list. The attribute will be highlighted and there will be a green checkmark to the left of it. This green checkmark identifies the current data attribute.

Pop-up windows are used to display the value of the current attribute at a particular point or section of an object (see "Data Probes" under Tools, on p. 83, for information about pop-ups). By default, the pop-ups display only the current attribute. The value associated with the current attribute is located to the right of the word Value in the popup window.

To view the value of all attributes for an object in the pop-up windows, choose **Extended Popup Info** from the **Display** tab of the **Properties** dialog of the view object. An example of a view window displaying an ArcView shape file, an extended popup window, the properties dialog for the shape file (data item) and the properties dialog for the 2D View object is shown below.

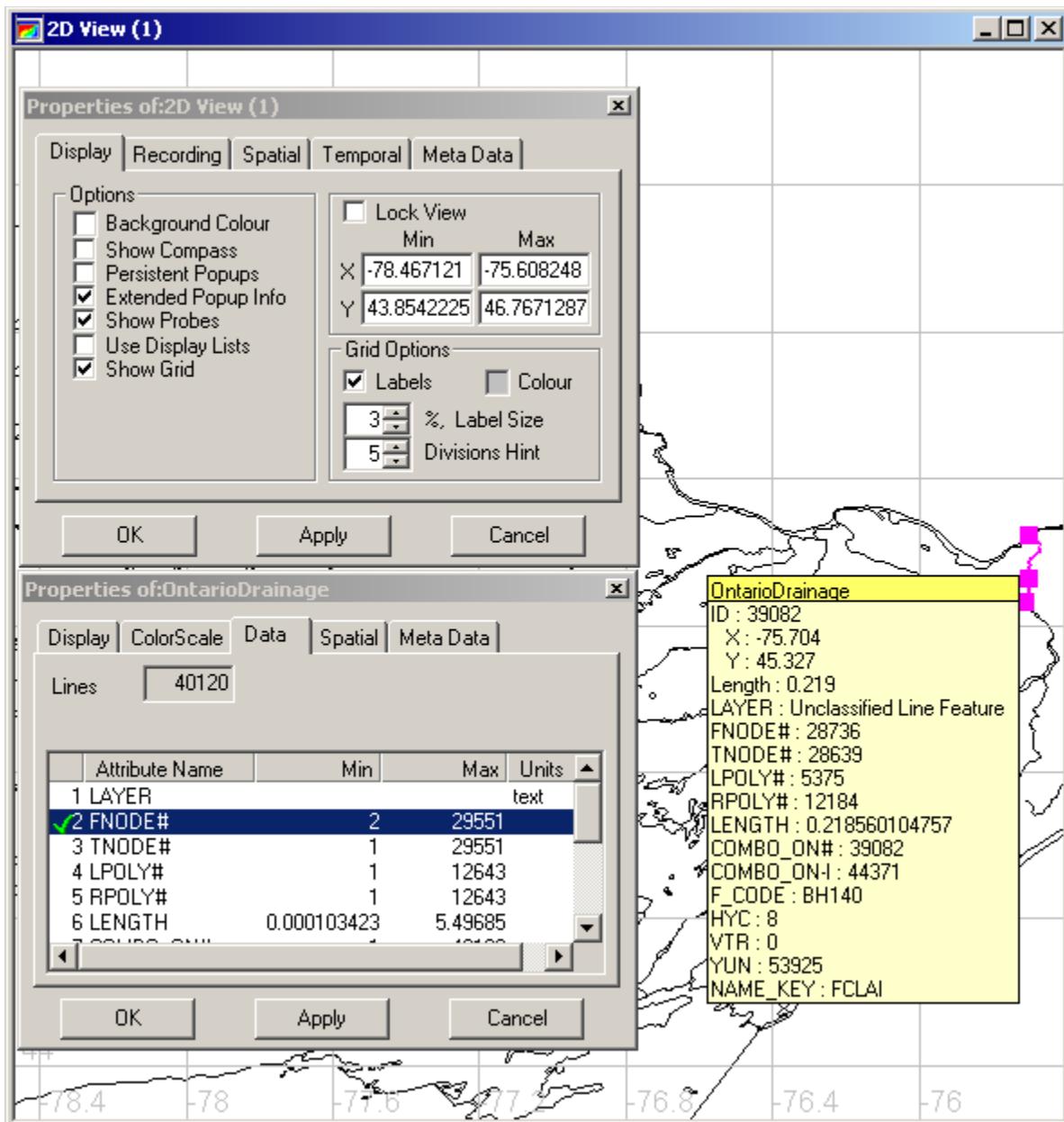


Figure 1.9: The popup window displays useful data about the current attribute

Note: The **Length:** displayed in the upper portion of the popup window, 0.219, is the length of the selected section of the river as calculated by EnSim. The **LENGTH:** attribute displayed in the lower portion of the popup window, 0.218560104757, was specified in the attributes of the shape file.

For an object being displayed in 1D, such as a time series, the data attributes displayed in popup windows are slightly different from those displayed for objects in 2D or 3D. A sample popup for data point of a time series in 1D is shown below. The time series was extracted from a T3 mesh.

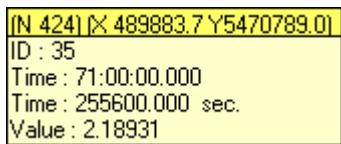


Figure 1.10: A 1D Time Series popup

- The title bar (the darker yellow bar at the top of the popup) shows from where on the mesh the time series was extracted, at node 424, and the location of this node.
- **ID** refers to the order of the data point in the time series. This point was the 35th data point in the data series.
- **Time**: There are two **Time**: values in a 1D pop-up window. The first is the time displayed in hours:minutes:seconds.decimal seconds. The second **Time**: is the same time displayed in decimal seconds. This data point was recorded at 71 hours, or 255,600 seconds.
- **Value**: This is the measured quantity, as displayed on the y-axis. This time series was extracted (see "Extracting Time Series" under Extracting Data, on p. 96) from a model of water depth over time. After 71 hours of simulated time, the depth of the water was 2.18931 m.

1.4.3.4 Spatial

The **Spatial** tab shows the x- and y-extents of the object. The units of the extents (distances and locations) depend on those used in the source file (i.e. if all data in the source file are in units of feet, then the values shown in the Spatial Tab will have units of feet). This tab is not available for 1D data or objects that contain other objects. An example of the **Spatial** tab is shown below.

There are two main sections on the **Spatial** tab. The **Attributes** section describes the origin, node count, delta, extent, and angle of the data item. The **Coordinate System** section describes the projection and datum of the coordinate system. Both of these sections are discussed further in the following sections. This page varies for different object types. Following these sections is a list of criteria for coordinate system selection.

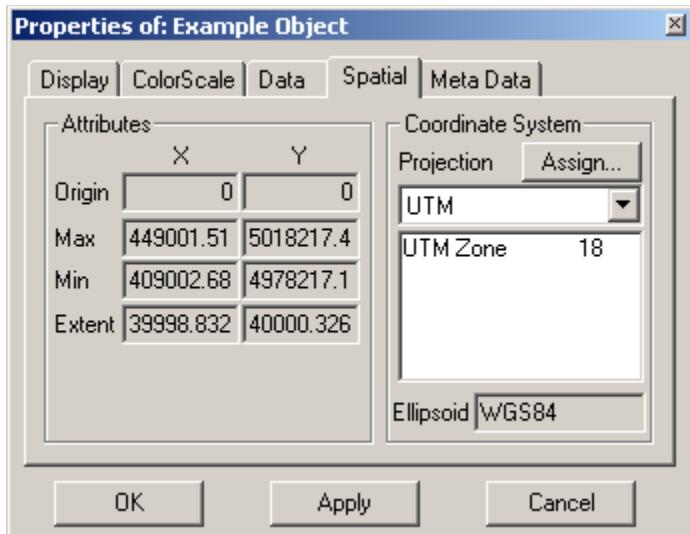


Figure 1.11: The Spatial tab of the Properties dialog has two sections

1.4.3.4.1 Attributes

The attributes shown in this section will vary from one object type to another. The possible attributes shown here are:

- **Origin:** This is the location of the southwest corner of a grid object in the assigned coordinate system.
- **Max.:** This is the location of the most extreme top right point (Northeast) of the object.
- **Min.:** This is the location of the most extreme bottom left point (Southwest) of the object.
- **Extent:** Under the X column, this is the distance in the x-direction between the Min. and Max. points of the object. Under the Y column, this is the distance in the y-direction between the Min. and Max. points of the object.
- **Nodes:** This is the number of nodes in the X & Y directions of a grid object.
- **Delta:** This is the distance between each node.

For a GeoTIFF, the attributes shown represent the X and Y coordinates of each corner of the object.

1.4.3.4.2 Coordinate Systems

All spatial EnSim Objects have a sense of coordinate system. Keywords identifying the coordinate system are found in the file header. The coordinate systems recognized by EnSim are LatLong, UTM, MTM, Polar Stereographic, Lambert Conformal, Albers, and Cartesian. The default coordinate system for any object is Cartesian. Some imported objects contain coordinate system information (e.g. DEM, MapInfo *.mif).

1.4.3.4.3 Coordinate System - Converting Projections

To change the projection of the object:



Figure 1.12: The Coordinate System projection can be changed within the Spatial tab

1. Select the projection from the **Projection** list box in the Spatial tab. The projections available are LatLong, UTM, MTM, Polar Stereographic, Lambert Conformal, and Albers.
 - If LatLong, UTM, MTM, Polar Stereographic, Lambert Conformal, or Albers has been selected, the central edit box will become active. Enter the appropriate details.
 - The Ellipsoid (Datum) list box will appear greyed out as Datum shifts are not allowed.
2. Select the **Apply** button. The object will now appear in the view with the new projection.

1.4.3.4.4 Coordinate Systems - Assigning Projections

To assign a coordinate system to an object:

1. Select the **Assign...** button.
2. The following dialog box will appear:

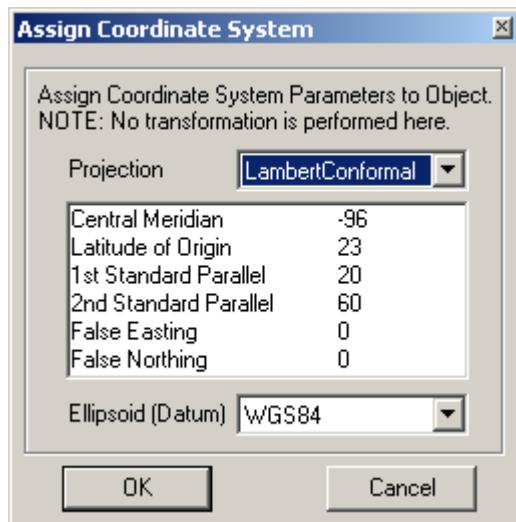


Figure 1.13: The Assign Coordinate System dialog box allows for seven possible projections

NOTE: As noted at the top of this dialog, this option is used strictly to assign a coordinate system to the data item. It does NOT perform coordinate transformations.

3. Select the projection from the **Projection** list box. The projections available are LatLong, UTM, MTM, Polar Stereographic, Lambert Conformal, Albers, and Cartesian.
4. If LatLong, UTM, MTM, Polar Stereographic, Lambert Conformal or Albers has been selected, the central edit box will become active. Enter the appropriate details:
 - **LatLong:** Select **-180 to 180 or 0 to 360** degrees.
 - **UTM or MTM:** Enter the zone number.
 - **Polar Stereographic:** Enter the **Centre Latitude** and **Centre Longitude**.
 - **Lambert Conformal or Albers:** Enter the **Central Meridian**, **Latitude of Origin**, **1st Standard Parallel**, **2nd Standard Parallel**, **False Easting** and **False Northing**.
5. Select an appropriate ellipsoid from the **Ellipsoid (Datum)** list box. The available ellipsoids are WGS84, WGS72, GRS80 (NAD83), Clark 1866 (NAD 27), and Sphere.

See "Ellipsoids", on p. 28 for more information.

6. Select the **OK** button to confirm your selections.
7. If a mistake has been made in assigning the coordinate system, reselect the **Assign...** button on the Spatial tab.

1.4.3.4.5 Ellipsoids

EnSim lets you quickly and easily convert spatial data between projections (e.g., from LatLong to UTM). However, to transform the coordinate data to the new projection, it is essential the data object is assigned the correct ellipsoid. The table below shows the ellipsoids supported by EnSim and their associated parameters.

Ellipsoid	Earth's Radius (m)	Flattening
WGS84	6,378,137.0	1.0 / 298.257223563
WGS72	6,378,135.0	1.0 / 298.26
GRS80 (NAD83)	6,378,137.0	1.0 / 298.257222101
Clarke1866 (NAD27)	6,378,206.4	1.0 / 294.9786982
Sphere	6,371,000.0	0.0 (perfect sphere)

The cause of misaligned data within a view is most likely non-matching ellipsoids. For example, overlaying NAD27 and NAD83 spatial data may produce an offset of up to 200 metres.

If the projection and or the ellipsoid are undefined, EnSim assumes that the data will use the default Cartesian coordinate system. With respect to LatLong projected data, there may be some confusion with regards to the defined ellipsoid. EnSim expects the LatLong data object

to have an ellipsoid/datum assigned. The sphere is a valid ellipsoid. Software packages such as MapInfo and ESRI's ArcInfo/ArcView act differently when loading LatLong data with undefined ellipsoids. ESRI assumes the data is mapped to a sphere, while MapInfo leaves it undefined. With MapInfo, you should be aware that reprojecting LatLong data with an undefined ellipsoid to a coordinate system with a known ellipsoid simply assigns the ellipsoid. (i.e., LatLong data with an undefined ellipsoid projected to LatLong with the NAD83 ellipsoid assigns the ellipsoid to NAD83).

It is important to know the source ellipsoid of your imported data, even if it is not identified in the header of the file, and assign it within EnSim.

Note: EnSim does NOT perform datum conversion (e.g., from NAD27 to NAD83).

1.4.3.4.6 Selecting a Coordinate System

It is important to understand the connection between an object's coordinate system and the view in which the object has been placed. All objects that have not had a coordinate system assigned to them are assumed to use the Cartesian coordinate system, with an unknown ellipsoid. Refer to the previous section on assigning a coordinate system to an object for further details. The following section outlines the criteria for keeping track of a coordinate system for an object and a view.

- **Data Item Coordinate System Unknown, View Coordinate System Unknown**

The data item can be dragged onto the view.

- **Data Item Coordinate System Known, View Coordinate System Unknown**

If the data item has a known coordinate system, once the object has been placed in the view, the view will be assigned the coordinate system of the data item.

- **Data Item Coordinate System Unknown, View Coordinate System Known**

If a view already has a coordinate system from another object, the view will give a warning when you attempt to view an object with an unknown coordinate system. An error message similar to the dialog shown below will appear. To avoid this problem, assign a coordinate system to the object. Refer to the previous section for further details.

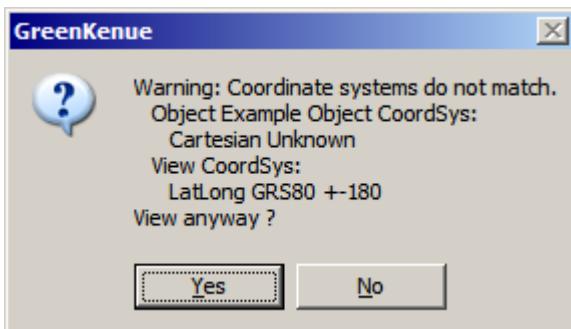


Figure 1.14: This error message is shown when viewing an object with an unknown coordinate system in a view with a known coordinate system

- **Data Item Coordinate System Known, View Coordinate System Identical**

If both the data item and the view have identical coordinate systems, the object can be dragged onto the view.

- **Data Item Coordinate System Known, View Coordinate System Different**

If the data item has a coordinate system different from the view, a warning message similar to the dialog shown below will appear.

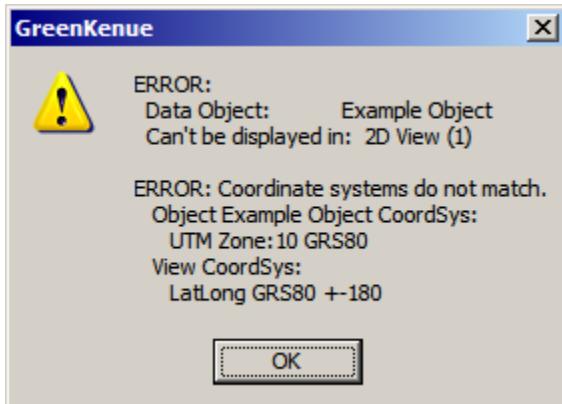


Figure 1.15: This error message is shown when an object's coordinate system does not match that of its view

- **New Data Item Created in View**

All new data items created in a view will acquire the coordinate system of the view.

- **Data Item Extracted from Data Item with Existing Coordinate System**

Data items extracted from other objects items will acquire the coordinate system of their parent object.

- **Multiple Data Items in View**

Changing the coordinate system of a data item in a view with multiple data items displayed will not change the coordinate system of the view. To view the object that has had its coordinate system changed, move it to a new view or change it back to the coordinate system of the view.

If only one object is in the view, then the view's coordinate system is changed when the object's coordinate system is changed.

Similarly, if multiple data items are displayed in a particular view, the view's coordinate system will change only when all of the data items have been changed to a new coordinate system.

For example, if two data items, both with the Polar Stereographic coordinate system, are displayed in a Polar Stereographic view, the view's coordinate system will remain the same if one of the data items is changed to the Lat/Long coordinate system. Only when the second data item is also changed to the Lat/Long coordinate system will the view's coordinate system change to match.

1.4.3.5 Meta Data

The **Meta Data** tab displays the information documented for the data (e.g. source, file type), obtained from the file header (see "File Headers", on p. 264, for information on file headers). A sample **Meta Data** tab is shown below.

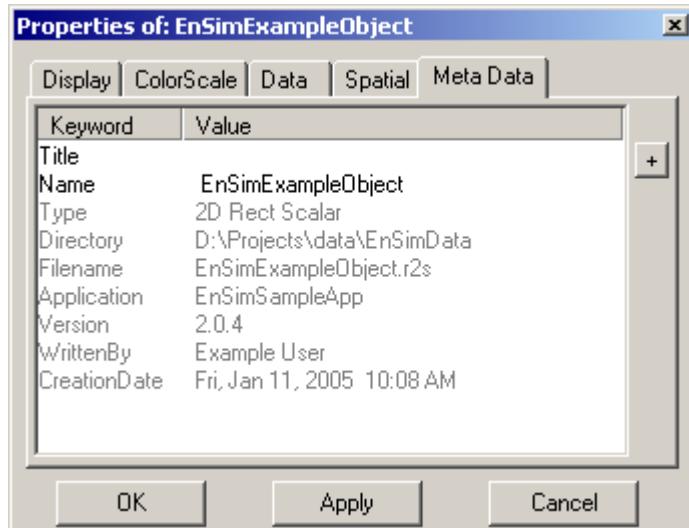


Figure 1.16: The Meta Data tab shows information about the object file

Some meta data fields may be changed (depending on the object). For example, the name and title of an object may be changed from this tab. Changing the **Title** field alters the title appearing in the header of the Properties dialog. Changing the **Name** field alters the name of the object that appears in the WorkSpace. Arbitrary keyword values may be added using the **+** button. The new keywords will be written to the standard EnSim file header. See "File Headers", on p. 264, for more information about keywords and file headers.

Read-only fields are greyed out.

1.4.3.6 Applying Changes to an Object's Properties

When the parameters in the Properties dialog are satisfactory, press **Apply** to apply the new parameters to the object and keep the Properties dialog open, or press **OK** to apply the parameters and close the Properties dialog.

1.4.3.7 Copying Data Item Properties

Display properties and colour scales of data items can be copied and applied to other data items. The steps are as follows:

To copy data item properties:

1. Select the data item with the desired display style and colour scale.
2. Select the **Copy Display Style** command from the **Edit** menu.
3. Select the data item to which the display style and colour scale are to be applied.

4. Select the **Paste Display Style** command from the **Edit** menu.

All properties on the **Display Style** and **Colour Scale** tabs will be applied to the data item, with the following exceptions:

- **Style** (wireframe, surface, etc.) will not be copied if the objects are of different types and have different possible styles. For example, a 2D line set and a rectangular grid have no styles in common. As a result, styles cannot be copied from one to the other. See "Display Properties" under Properties of Data Items, on p. 17, for more information.
- **Animate** checkbox settings (see "Animation", on p. 61).
- **Show** and **Options** settings from the Legend section of the Colour Scale tab (see "Colour Scale", on p. 20).

1.5 VIEWS

Views are windows within EnSim where data items can be displayed. There are six types of views: 1-dimensional (1D), Polar, 2-dimensional (2D), 3-dimensional (3D), Spherical, and the Report view, each with unique display properties. Within views, animation of time-varying data is controlled, data can be recorded in movie (*.avi) format, and various characteristics of the display can be altered, including the background colour, 2D grid display and extended popup information.

View windows are treated as objects in the WorkSpace. View objects have properties, much like data items, that affect the environment in which data items are displayed.

Relationships between data items and view windows are generally handled in the workspace. For example, changing the display parameters of an object in a view, re-layering multiple objects in a 2D view, and adding and removing objects to and from views are all performed from the WorkSpace.

1.5.1 Creating a View Window

New view windows are added to the WorkSpace using one of the view window buttons, or by using one of the **New View** menu commands in the **Window** menu: 1D, polar, 2D, 3D, spherical, or report. The 1D button is , the polar button is , the 2D button is , the 3D button is , the spherical button is , and the report button is .

Views are automatically numbered according to the order in which they are created. The numbers are not related to view type, and can be changed by editing the title or subtitle in the **Meta Data** tab of the view's Properties dialog box.

Multiple view windows may be opened and displayed simultaneously. There are three options for fitting the windows automatically to the available space: **Cascade**, **Tile Horizontally** and **Tile Vertically**. These are found in the **Windows** menu.

1.5.2 Removing a View Window

View windows are deleted from the screen by using the  button in the top right hand corner of the view window.

Data items can be removed from a view by selecting the data item and using the <Delete> key, selecting **Remove** from its shortcut menu, or by selecting **Edit→Remove** from the menu bar.

1.5.3 Properties Shared by all View Types

The view window's Properties dialog box can be accessed in three ways:

- With the view window selected, choose the **Properties** command from the **Edit** menu.
- With the view window selected, choose the **Properties** command from the shortcut menu.

- Double-click on the view window object in the WorkSpace.

1.5.3.1 The Properties Dialog

A view window's Properties dialog has as a minimum a **Display** and **Meta Data** tab. Other tabs may or may not be available. The Display tabs differ considerably for each view type and are discussed in the sections pertaining to the specific view type. The other tabs are the same for each view type and are discussed briefly below in this section.

- **Display:** This tab varies according to the type of view. See the chapters specific to each view type:
 - 1D View Window Display Properties (p. 36)
 - Polar View Window Display Properties (p. 39)
 - 2D View Window Display Properties (p. 42)
 - 3D View Window Display Properties (p. 45)
 - Spherical View Window Display Properties (p. 48)
 - Report View Window Page Setup Properties (p. 52)
- **Recording:** Interface for creating recordings of animated data. For more information, see the section "Recording" under Saving and Copying Images, on p. 65.
- **Spatial:** Displays the x-, y-, and z-spatial extent of the window. The extents are automatically determined from the data items currently in view.
- **Temporal:** Allows you to create a clock decoration object and to adjust the target frame rate for interactive playback of an animation. The frame rate is the number of frames per second (fps). See the section "The Simulation Clock" under View Decorations, on p. 58, and the section "Animation" under Views, on p. 61 for more information.
- **Tools:** Currently only available for the 2D view. This tab allows you to modify live stream line cursor settings (see section "The Live Stream Lines Cursor" under Tools, on p. 86 for more information).
- **Meta Data:** Provides information regarding the type of window. By clicking in the title field, you may change the title of the view in the WorkSpace and the title block of the view window's Properties dialog. This also changes the title of the view when it is printed. See the section Printing under "Saving and Copying Images."

Press  to update changes made to the controls in the tabs and keep the Properties dialog box open. Press  to update the changes and close the dialog. Either selection will update changes made on all tabs.

1.5.4 The 1D View Window

The *1D view window* can display either time series, XY data items or a 3D Line Set in a graph format. The 1D view display properties can be edited, and objects can be manipulated in the 1D view. A new 1D view window can be opened by pressing the  button in the tool bar.

A typical 1D view window is displayed below.

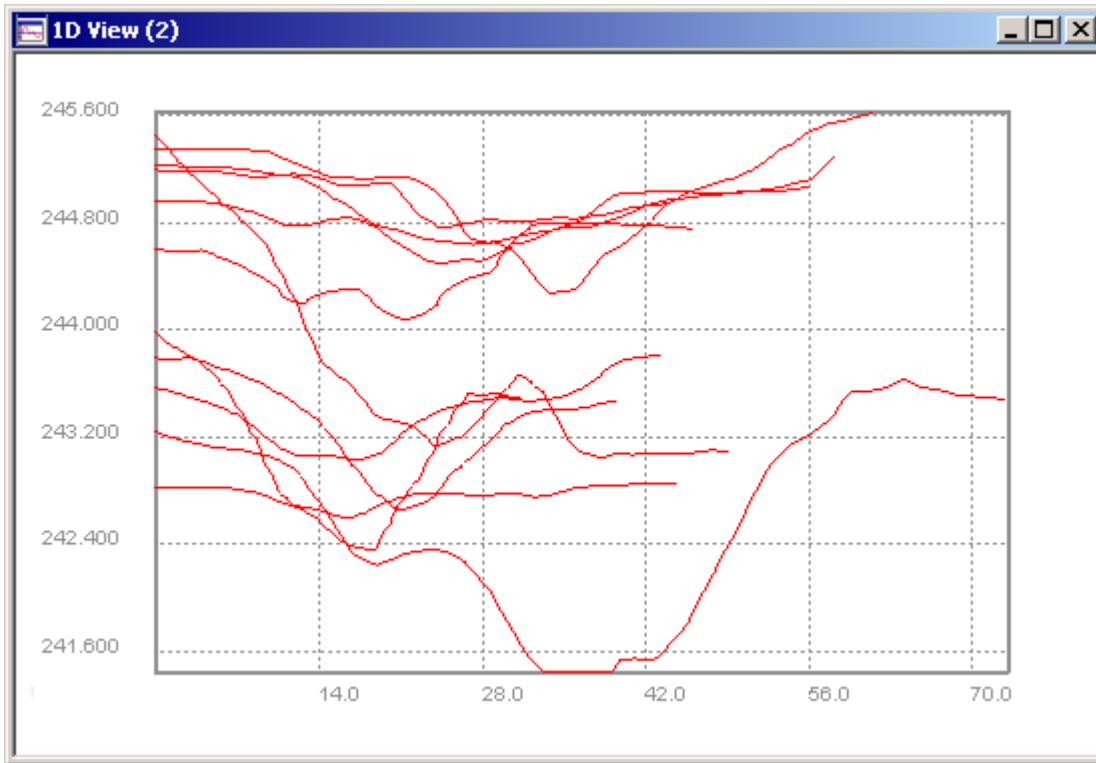


Figure 1.17: A typical 1D view window displays changes in value over time

1.5.4.1 Labels of Axes in a 1D View

For a time series with an explicit date, the date will appear on the axis. See "Time Series [ts1 / ts2 / ts3 / ts4 / ts5]", on p. 288, for details concerning time step formatting. For a 3D line set in the 1D view, the units are distance along the line from the starting point of the line set. For an XY data item, the axes take on the units of each attribute represented in the graph.

The x-axis and y-axis (the value axis) are not labelled automatically, as multiple time series that are not expressed in the same units may be displayed in the same 1D View. A label can be created by clicking the  button in the tool bar (see "Labels" under View Decorations, on p. 59, for details). The units of the data may be found in a popup window (see "Data Probes" under Tools, on p. 83), or on the Data tab of the Properties dialog.

1.5.4.2 The 1D View Window Status Bar

The bottom of the EnSim application window provides information on the current window. For an active 1D window with a time series, the location of the cursor is displayed with respect to the time axis (T) and the value axis (or y-axis) (V). For an active 1D window with a line set, the location of the cursor is displayed with respect to the distance axis (D) and the value axis (or y-axis) (V).



1.5.4.3 Manipulating the 1D View

The view can be panned by dragging the mouse with the left mouse button depressed. Zoom in by pressing the **<Ctrl>** key while dragging the mouse upwards, or by moving the mouse wheel up, if that option is available. Zoom out by pressing the **<Ctrl>** key while dragging the mouse downwards or by moving the mouse wheel down. While the view is being manipulated, a hand cursor  will appear.

The view can also be manipulated by adjusting the **X** and **Y** minimum and maximum extents in the **Display** tab of the view's **Properties** dialog box. These will be the minimum and maximum values displayed on the **X** and **Y** axes.

An infinite number of moves can be undone by the **Undo Move** command in the shortcut menu of the view object. The **Default View** command in the view's shortcut menu allows you to return to the default view, which centres the entire object in the view window.

1.5.4.4 Display Properties of the 1D View Window

The display properties of the 1D window are changed in the **Display** tab of the view's **Properties** dialog box. A sample Display tab for a 1D window is shown below.

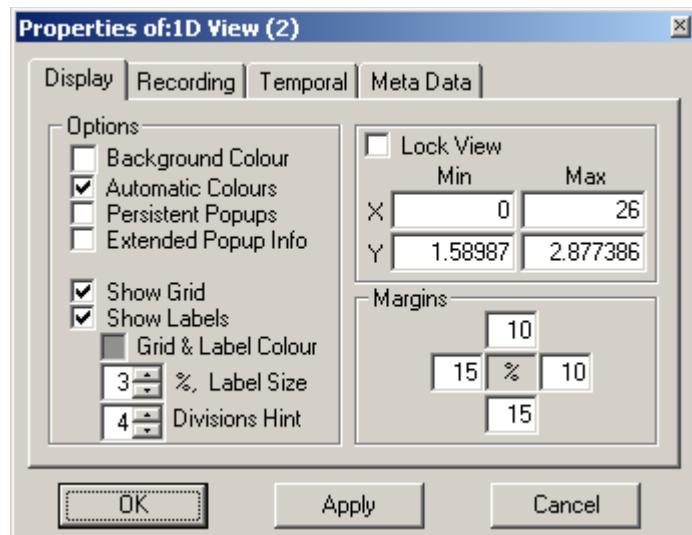


Figure 1.18: The Display Properties dialog of a 1D view

- **Background Colour:** The box is not a checkbox, but a colour selector indicating the colour to be applied to the background of the 1D View window. Upon selecting the box, a colour selection dialog appears. The box will display the colour selected.
- **Automatic Colours:** If this box is checked, a different colour will automatically be assigned to each data object that is added to the 1D View window. This is especially useful when comparing two or more timeseries.
- **Persistent Popups, Extended Popup Info:** These control the view's data probes. See the section on "Data Probes" under Tools, on p. 83.

- **Show Grid:** When the check box is turned off, the grid lines are removed. The axes' coordinates remain visible.
- **Show Labels:** This check box toggles the axes' coordinate labels.
- **Grid & Label Colour:** This is not a checkbox, but a colour selector indicating the colour to be applied to the grid and the axis-interval labels. When the box is selected, a colour selection dialog appears. The box will display the colour selected.
- **Label Size:** This controls the size of the numbers along the axes. The values represent the percentage of the view window size.
- **Divisions Hint:** Enter the number of grid divisions to be displayed in the horizontal direction. The maximum number of divisions that can be entered here is 8. As the number of divisions is dependent on the size of the window and the data displayed, this parameter is used as a guideline for the number of divisions.
- **Lock View:** When toggled on, the ability to pan the view or zoom in or out will be disabled. View decoration objects can still be moved. When the view is locked, the green padlock  in the bottom right-hand corner of the EnSim window turns red .
- **X and Y:** These are the current extents of the view along the respective axes.
- **Margins:** This controls the white space surrounding the graph as a percentage of the view window.

1.5.5 The Polar View Window

The *polar view window* can display either a velocity rose or a vector time series in a polar plot format. The polar view display properties can be edited, and objects can be manipulated in the polar view. A new polar view window can be opened by pressing the  button in the tool bar.

A typical polar view window is displayed below.

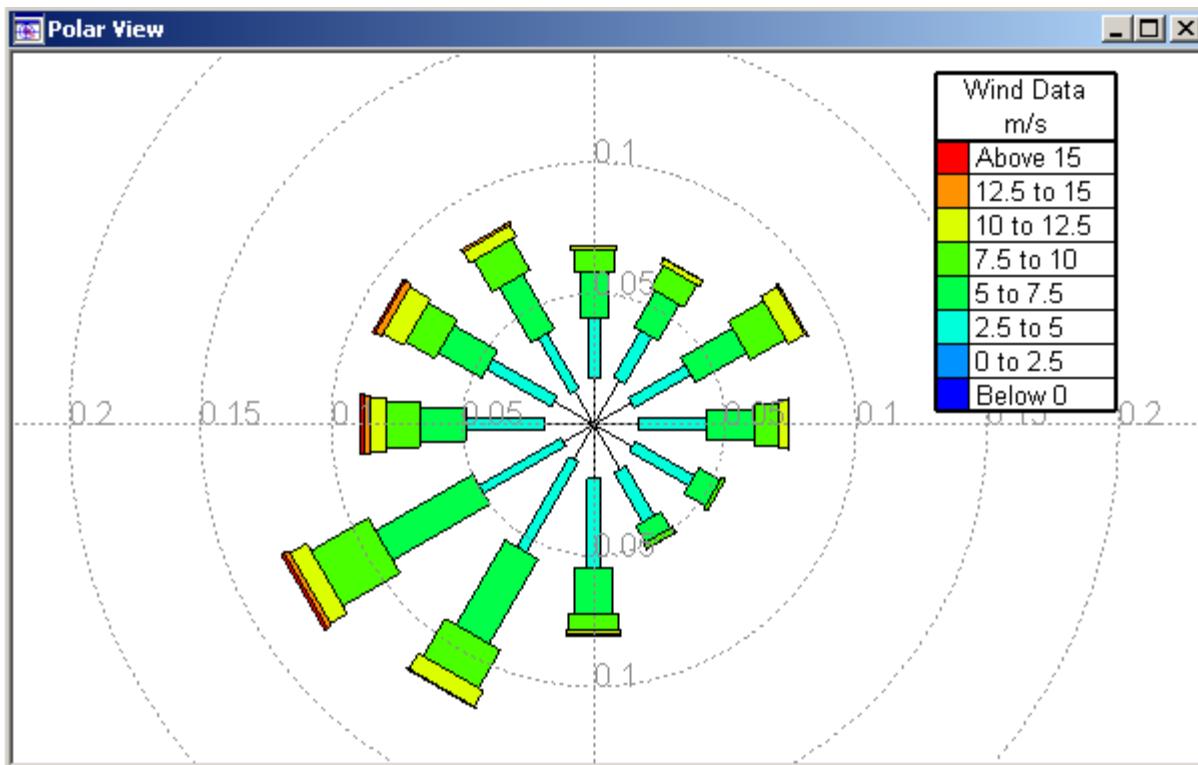


Figure 1.19: A typical polar view window displaying probabilities of wind speed against direction

1.5.5.1 Coordinates in a Polar View

Each point within the polar view is determined by a magnitude and an angle. The magnitude is measured radially from the centre of the plot. The direction spans clockwise from 0 to 360 degrees starting with 0 degrees on the upward vertical axis (positive y-axis).

1.5.5.2 The Polar View Window Status Bar

The bottom of the EnSim application window provides information on the current window. For an active polar window with a time series, the location of the cursor is displayed with respect to the magnitude and direction.



1.5.5.3 Manipulating the Polar View

The view can be panned by dragging the mouse with the left mouse button depressed. Zoom in by pressing the **<Ctrl>** key while dragging the mouse upwards, or by moving the mouse wheel up, if that option is available. Zoom out by pressing the **<Ctrl>** key while dragging the mouse downwards or by moving the mouse wheel down. While the view is being manipulated, a hand cursor  will appear.

The view can also be manipulated by adjusting the **X** and **Y** minimum and maximum extents in the **Display** tab of the view's **Properties** dialog box. These will be the minimum and maximum values displayed on the **X** and **Y** axes.

An infinite number of moves can be undone by the **Undo Move** command in the shortcut menu of the view object. The **Default View** command in the view's shortcut menu allows you to return to the default view, which centres the entire object in the view window.

1.5.5.4 Display Properties of the Polar View Window

The display properties of the polar window are changed in the **Display** tab of the view's **Properties** dialog box. A sample Display tab for a polar window is shown below.

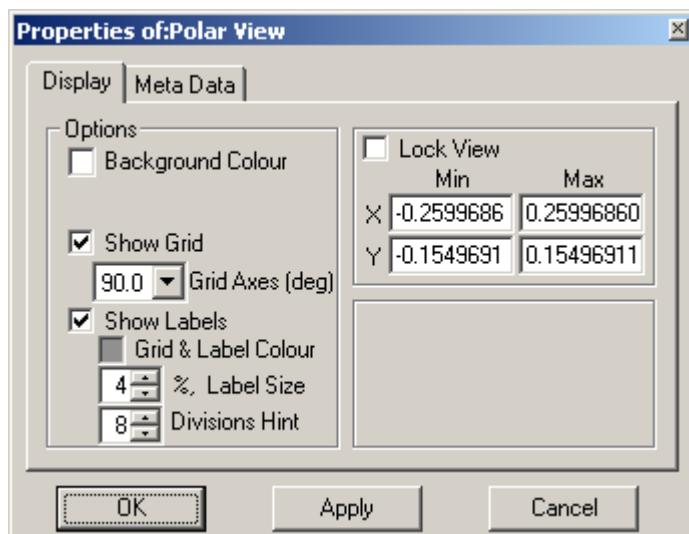


Figure 1.20: The Display Properties dialog of a polar view

- **Background Colour:** The box is not a checkbox, but a colour selector indicating the colour to be applied to the background of the Polar View window. Upon selecting the box, a colour selection dialog appears. The box will display the colour selected.
- **Show Grid:** When the check box is turned off, the grid lines are removed. The axes' coordinates remain visible.
- **Grid Axes (deg):** This defines the spacing of radial axes displayed in the view. Axes may be drawn at 22.5, 30, 45, or 90 degree intervals.
- **Show Labels:** This check box toggles the axes' coordinate labels.
- **Grid & Label Colour:** This is a colour selector indicating the colour to be applied to the grid and the axis-interval labels. When the box is selected, a colour selection dialog appears. The box will display the colour selected.
- **Label Size:** This controls the size of the numbers along the axes. The values represent the percentage of the view window size.
- **Divisions Hint:** Enter the number of grid divisions to be displayed in the horizontal direction. The maximum number of divisions that can be entered here is 8. As the number of divisions

is dependent on the size of the window and the data displayed, this parameter is used as a guideline for the number of divisions.

- **Lock View:** When toggled on, the ability to pan the view or zoom in or out will be disabled. View decoration objects can still be moved. When the view is locked, the green padlock  in the bottom right-hand corner of the EnSim window turns red .
- **X and Y:** These are the current extents of the view along the respective axes.

1.5.6 The 2D View Window

The *2D view window* displays grid and cartographic data in a plane view. The default view shows the entire extent of the data within the view, with an overlaying grid and coordinates. The 2D view display properties can be edited, and objects can be manipulated in the 2D view. A new 2D View window can be opened by pressing the  button.

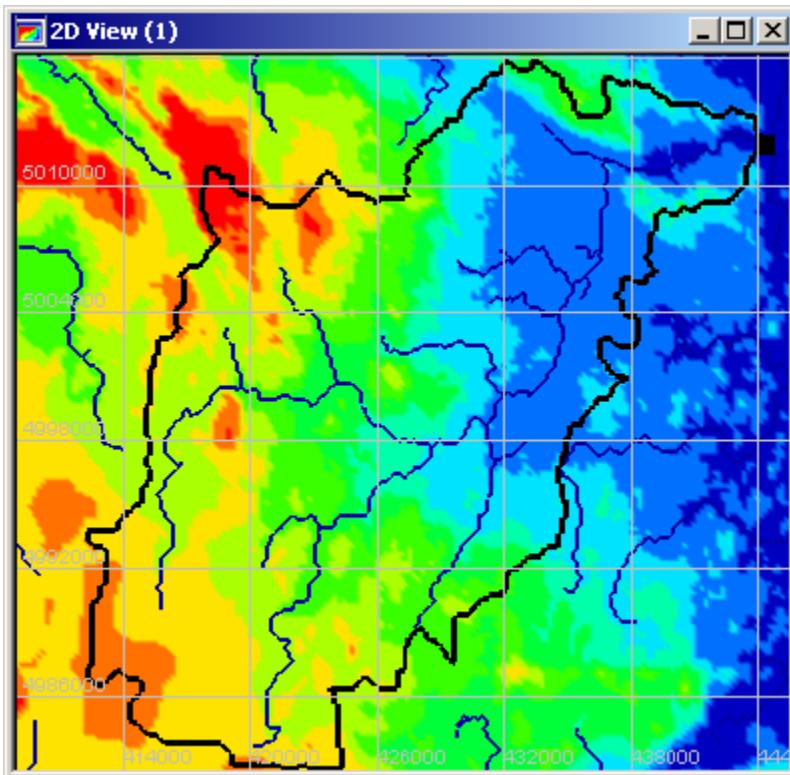


Figure 1.21: This 2D View window shows a watershed from Green Kenu

1.5.6.1 Coordinate Systems and Units in 2D Views

The default coordinate system used in the 2D view is Cartesian, but other systems may also be used. For example, if a data item is georeferenced in latitude and longitude coordinates, these coordinates are displayed along the axes of the grid system. If the data is georeferenced in a UTM projection, the eastings and northings are the numbers displayed on the axes. If the data is not georeferenced, the units of measure used to make the grid may be anything (e.g. feet, metres, miles, kilometres, etc.). You are advised to ensure that all data used during an EnSim

session are in units compatible with other files used during the same session and with other software applications. Refer to the section "Coordinate Systems" under Properties of Data Items, on p. 26 for additional information on coordinate systems.

1.5.6.2 The 2D Window Status Bar

The bottom of the EnSim window provides information on the open window. For an active 2D window, the view's current coordinate system and the location of the cursor is displayed.



1.5.6.3 Manipulating the 2D View

- The view can be panned by dragging the cursor with the left mouse button held down.

Zoom in by pressing the <Ctrl> key while dragging the mouse upwards, or by moving the mouse wheel up. Zoom out by pressing the <Ctrl> key while dragging the mouse downwards or by moving the mouse wheel down. While the view is being manipulated, a hand cursor  will appear.

The view can also be manipulated by adjusting the **X** and **Y** minimum and maximum extents in the **Display** tab of the view's **Properties** dialog box.

Moves can be undone by the **Undo Move** command in the view's shortcut menu. The **Default View** command in the view's shortcut menu returns to the default view.

- Data items are drawn in the view in the reverse order in which they are added to the view window. The last data item to be added to the view is the top layer of objects drawn in the view and the first in the list of objects contained in the view. It will obscure all previously added objects if they overlap.

To move a data item already in the view to the top layer:

In the WorkSpace, drag the data item back onto the view object, as shown in the following figure.

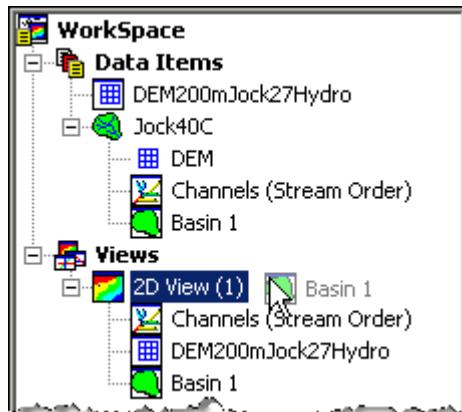


Figure 1.22: To move an object to the top layer, drag it back onto the view object

1.5.6.4 Display Properties of the 2D Window

The display properties of the 2D window are changed in the **Display** tab of the view's **Properties** dialog box.

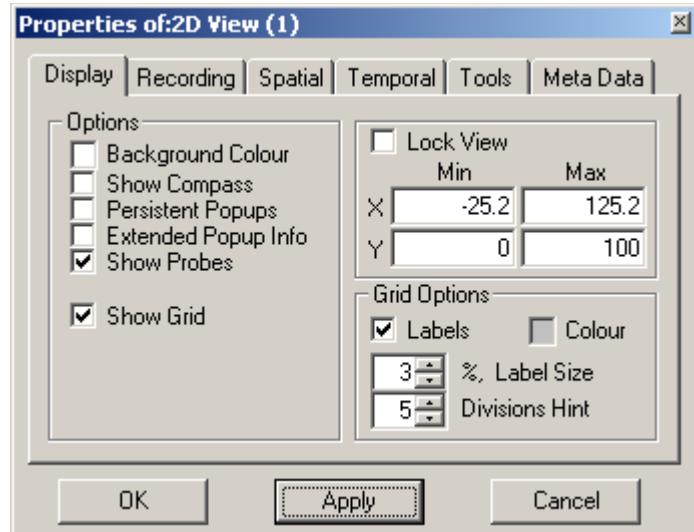


Figure 1.23: The Display Properties dialog of a 2D view

The display properties that can be edited include:

- **Background Colour:** The box is not a checkbox, but a colour selector indicating the colour to be applied to the background. Upon selecting the box, a colour selection dialog appears. The box will display the colour selected.
- **Show Compass:** The compass is a view decoration object, and is described in the section "The Compass" under View Decorations, on p. 58.
- **Persistent Popups, Extended Popup Info, Show Probes:** These control the view's data probes. See the section on "Data Probes" under Tools, on p. 83.
- **Show Grid:** When this checkbox is turned off, the lines of the grid are removed, but the coordinates remain visible. This is useful when viewing gridded data. The grid of the 2D View can sometimes overlap with the lines of the object, causing the lines to be hidden.
- **Lock View:** When toggled on, the ability to pan the view or zoom in or out will be disabled. View decoration objects can still be moved. When the view is locked, the green padlock  in the bottom right-hand corner of the EnSim window turns red .
- **X and Y:** These are the current extents of the view along the respective axes.
- **Labels:** This toggles the grid coordinate labels.
- **Colour:** This is a colour selector, not a checkbox. By clicking on the coloured square, a dialog appears, allowing you to choose a colour for the gridlines and the grid coordinates.
- **Label Size:** This controls the size of the numbers along the axes. The values represent the percentage of the view window size.

- **Divisions Hint:** Enter the number of grid divisions to be displayed in the horizontal direction. The maximum number of divisions that can be entered here is 8. As the number of divisions is dependent on the size of the window and the data displayed, this parameter is used as a guideline for the number of divisions.

1.5.7 The 3D View Window

The *3D view window* displays data in a perspective view. The default view shows a perspective from the negative x, negative y and positive z octant. The 3D view display properties can be edited, and objects can be manipulated in the 3D view. A new 3D View window can be opened by pressing the  button in the Tool bar. The sample 3D view shown below displays the same objects as the sample of the 2D view.

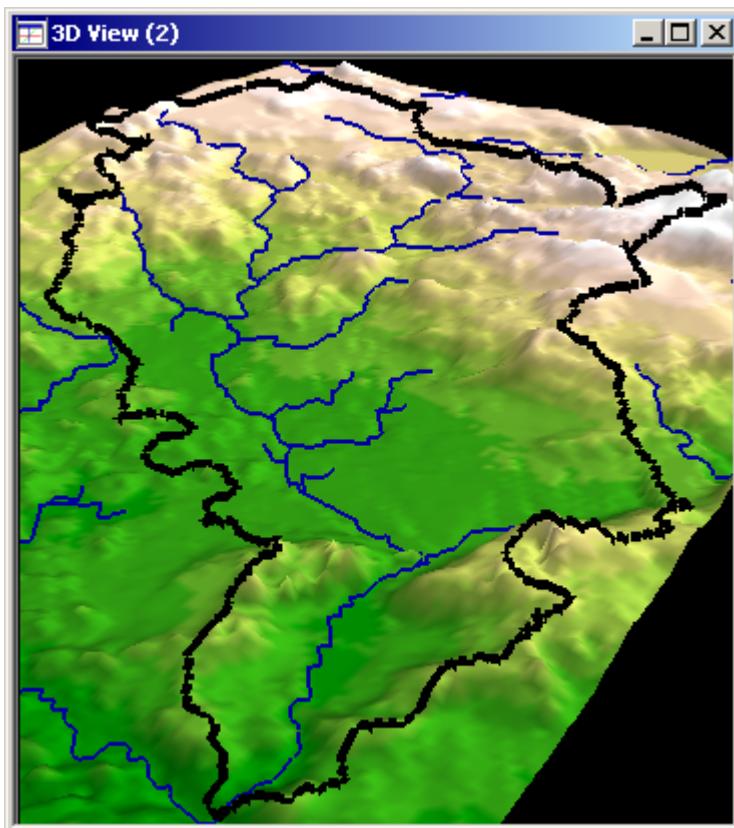
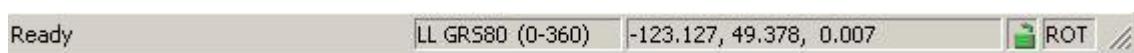


Figure 1.24: This 3D view shows the same data item as the 2D view on p. 36

1.5.7.1 The 3D Window Status Bar

The bottom of the EnSim window provides information on the open window. For the active 3D window, the view's current coordinate system and the location of the crosshairs is displayed, as well as the active type of view manipulation (ROT for rotation, TRN for translation).



1.5.7.2 Manipulating the 3D View

In EnSim 3D space, there are two ways in which the view can be manipulated:

- By dragging with the mouse, the view can be rotated or translated. Rotation and translation occur in the x-y plane, using the <Ctrl> key in conjunction with the mouse, or using the mouse wheel, rotation and translation can occur in the z or vertical direction. The options of **Rotate** or **Translate** can be chosen using commands in the shortcut menu or in the display tab of the view's **Properties** dialog box. During rotation manipulations, a hand with an arrow cursor  will appear. During translation manipulations, a hand cursor  will appear.

An infinite number of moves can be undone by the **Undo Move** command in the view's shortcut menu. The **Default View** command in the view's shortcut menu allows you to return to the default view.

- Changing the view parameters in the display tab of the view's **Properties** dialog box.

There are several types of controls to the view parameters

- **X, Y, and Z Camera:** Camera indicates the location in 3D space from which you are viewing the scene.
- **X, Y, and Z View Centre:** View centre refers to the location of the centre of the point of interest within a view. It is the centre of the view window. Note that the crosshair is always drawn at the centre of the view.
- **Field of View:** Analogous to the field of view of a camera lens, it is the angle that defines the limit of the size of area you see around the view centre. It has a zoom effect; decreasing the field of view is equivalent to zooming in, and increasing the field of view is equivalent to zooming out.

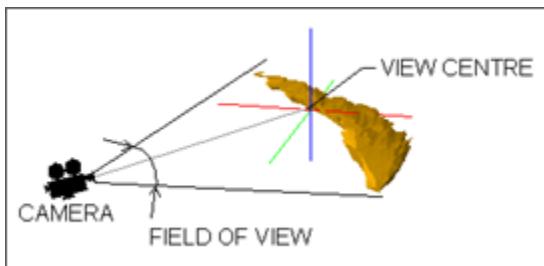


Figure 1.25: The Field of View determines the scope of the image produced

- **Near and Far:** *Near* and *far* are clipping planes of the view. Clipping planes are limits perpendicular to the line of sight between the camera and the view centre. By default, the clipping planes are located on either side of the view's data. In the view's coordinate system's units, near and far are defined by the distance of the clipping plane from the camera. If, while zooming in, parts of the image begin to disappear, the near parameter should be reduced in order to view all parts of the image.

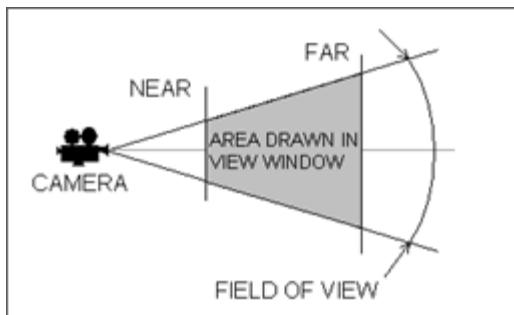


Figure 1.26: The Near and Far parameters determine the depth of the image

Moves cannot be undone if changed in the **Properties** dialog box. The **Default View** command in the view's shortcut menu allows you to return to the default view.

1.5.7.3 Display Properties of the 3D View Window

The display properties of the 3D window are changed in the **Display** tab of the view's **Properties** dialog box.

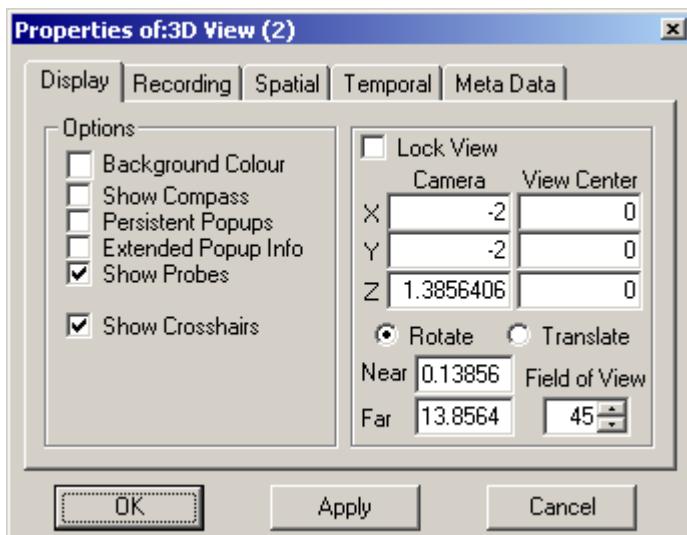


Figure 1.27: The Display Properties dialog of a 3D view

The display properties that can be edited include:

- **Background Colour:** The box is not a checkbox, but a colour selector indicating the colour to be applied to the background. Upon selecting the box, a colour selection dialog appears. The box will display the colour selected.
- **Show Compass:** The compass is a view decoration object, and is described in the section "The Compass" under View Decorations, on p. 58.
- **Persistent Popups, Extended Popup Info, Show Probes:** These control the view's data probes. See the section on "Data Probes" under Tools, on p. 83.
- **Show Crosshairs:** Crosshairs are the red, green, and blue axes defining the x, y, and z directions, respectively.

- **Lock View:** When toggled on, the ability to move data items in the view will be disabled. View decoration objects can still be moved. When the view is locked, the green padlock  in the bottom right-hand corner of the EnSim window turns red .
- **Rotate** and **Translate** control the movement of the viewing area, as described previously. See "Manipulating the 3D View", on p. 44, for more details.
- **X, Y, and Z extents, Near, Far, and Field of View** behave as described previously in this section. See "Manipulating the 3D View", on p. 44, for more details.

1.5.8 The Spherical View Window

The *spherical view window* displays data in a spherical view as though one was looking at the earth from space. The spherical view display properties can be edited, and objects can be manipulated in the spherical view. A new Spherical View window can be opened by pressing the  button in the Tool bar just as in all other views or select **New Sphere View** from the **Window** menu.

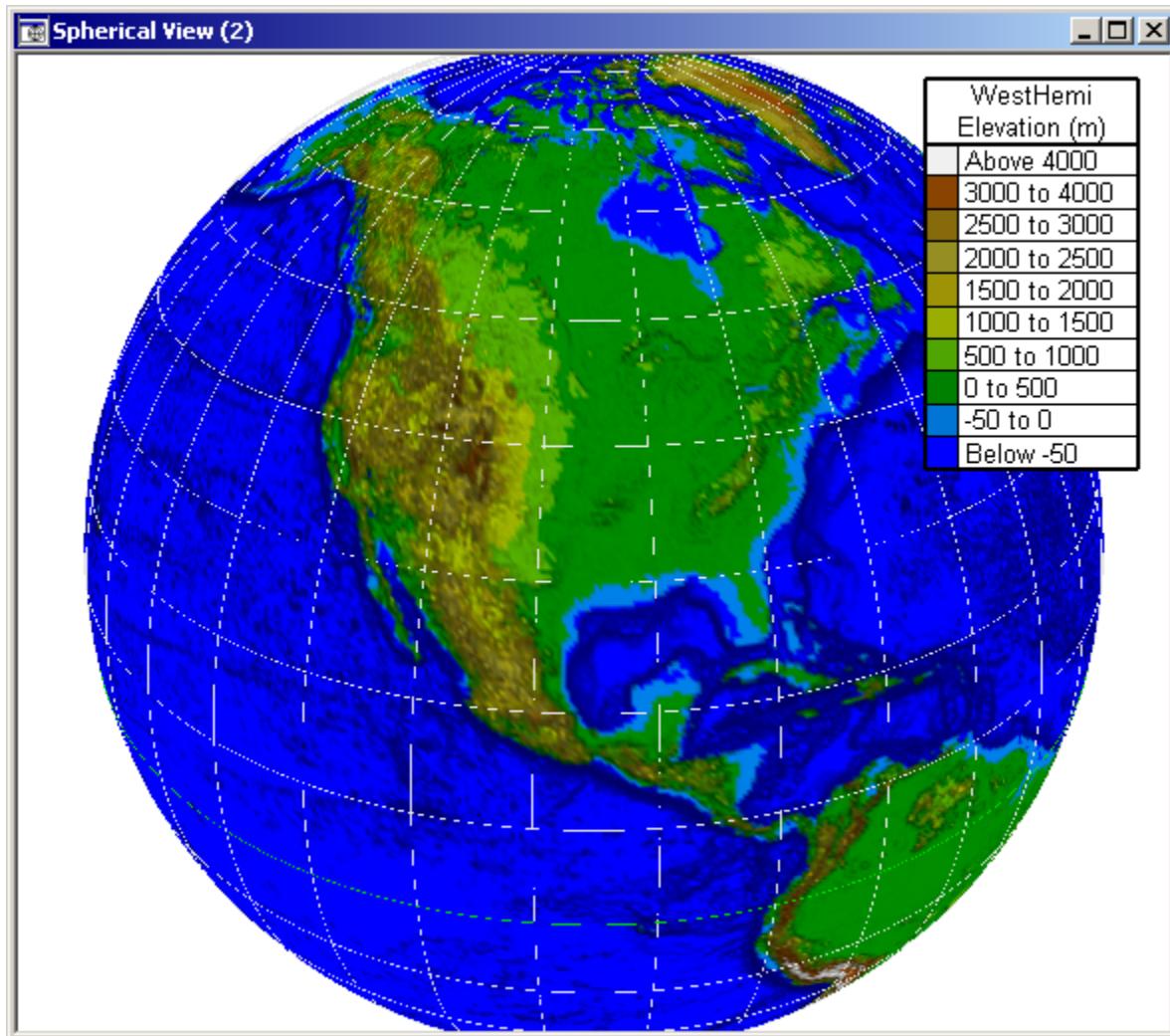


Figure 1.28: The Earth is most accurately seen in a spherical view

Note: Only data items containing LatLong coordinates may be viewed in a spherical view. Viewing objects in other coordinate systems, or changing a LatLong object to another system while it is being viewed in a spherical view, can have unpredictable effects.

1.5.8.1 The Spherical View Window Status Bar

The bottom of the EnSim window provides information on the open window. The status bar for the active spherical window is identical to the 3D view status bar, the view's current coordinate system and the location of the crosshairs is displayed, as well as the active type of view manipulation (ROT for rotation, TRN for translation).



1.5.8.2 Manipulating the Spherical View

In EnSim spherical space, there are two ways in which the view can be manipulated:

- By dragging with the mouse, the view can be rotated or translated. Rotation and translation occur from the surface of the object, using the **<Ctrl>** key in conjunction with the mouse. Using the mouse wheel, rotation and translation can occur in the z or vertical direction. The options of **Rotate** or **Translate** can be chosen using commands in the shortcut menu or in the display tab of the view's **Properties** dialog box. During rotation manipulations, a hand with an arrow cursor  will appear. During translation manipulations, a hand cursor  will appear.

An unlimited number of moves can be undone by the **Undo Move** command in the view's shortcut menu. The **Default View** command in the view's shortcut menu allows you to return to the default view.

- Changing the view parameters in the display tab of the view's **Properties** dialog box.

There are several types of controls to the view parameters

- **X, Y, and Z Camera:** Camera indicates the location in 3D space from which you are viewing the scene.
- **X, Y, and Z View Centre:** View centre refers to the location of the centre of the point of interest within a view. It is the centre of the view window. Note that the crosshair is always drawn at the centre of the view.
- **Field of View:** Analogous to the field of view of a camera lens, it is the angle that defines the limit of the size of area you see around the view centre. It has a zoom effect; decreasing the field of view is equivalent to zooming in, and increasing the field of view is equivalent to zooming out.
- **Near and Far:** *Near* and *far* are clipping planes of the view. Clipping planes are limits perpendicular to the line of sight between the camera and the view centre. By default, the clipping planes are located on either side of the view's data. In the view's coordinate system's units, near and far are defined by the distance of the clipping plane from the

camera. If, while zooming in, parts of the image begin to disappear, the near parameter should be reduced in order to view all parts of the image.

1.5.8.3 Display Properties of the Spherical View Window

The display properties of the spherical window are changed in the **Display** tab of the view's **Properties** dialog box.

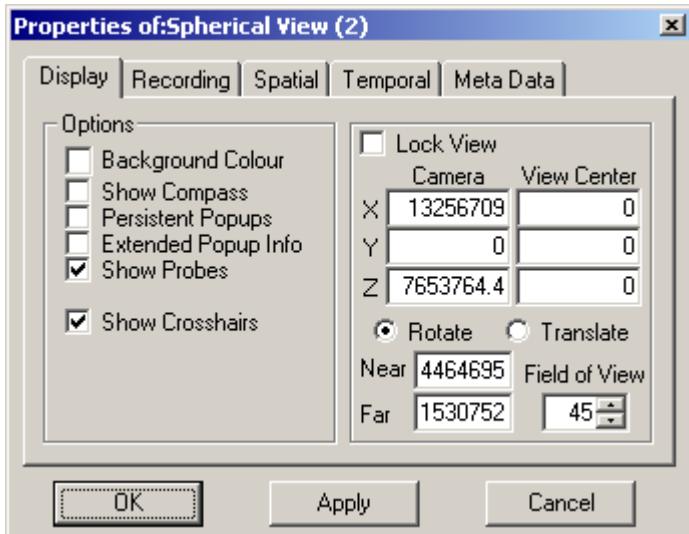


Figure 1.29: The Display Properties dialog of a spherical view

The display properties that can be edited include:

- **Background Colour:** The box is not a checkbox, but a colour selector indicating the colour to be applied to the background. Upon selecting the box, a colour selection dialog appears. The box will display the colour selected.
- **Show Compass:** The compass is a view decoration object, and is described in the section "The Compass" under View Decorations, on p. 58.
- **Persistent Popups, Extended Popup Info, Show Probes:** These control the view's data probes. See the section on "Data Probes" under Tools, on p. 83.
- **Show Crosshairs:** Crosshairs are the red, green, and blue axes defining the x, y, and z directions, respectively.
- **Lock View:** When toggled on, the ability to move data items within the view will be disabled. View decoration objects can still be moved. When the view is locked, the green padlock  in the bottom right-hand corner of the EnSim window turns red .
- **Rotate** and **Translate** control the movement of the viewing area, as described previously. See "Manipulating the Spherical View", on p. 47, for more details.
- **X, Y, and Z extents, Near, Far, and Field of View** behave as described previously in this section. See "Manipulating the Spherical View", on p. 47, for more details.

1.5.9 The Report View Window

The *report view window* allows you to display several views at once, and to synchronize several animated views. The report view also lets you prepare one or more views for printing, while including legends, headers, or other formatting objects. Unlike other views, the report view does not display data items. Instead, other types of views can be dragged onto and displayed within a report view. A new report view window can be opened by pressing the button, or by selecting **Window→New Report View**. You can open an already-formatted report template by selecting **Window→Load Template....** See "Report View Templates", on p. 53, for more information.

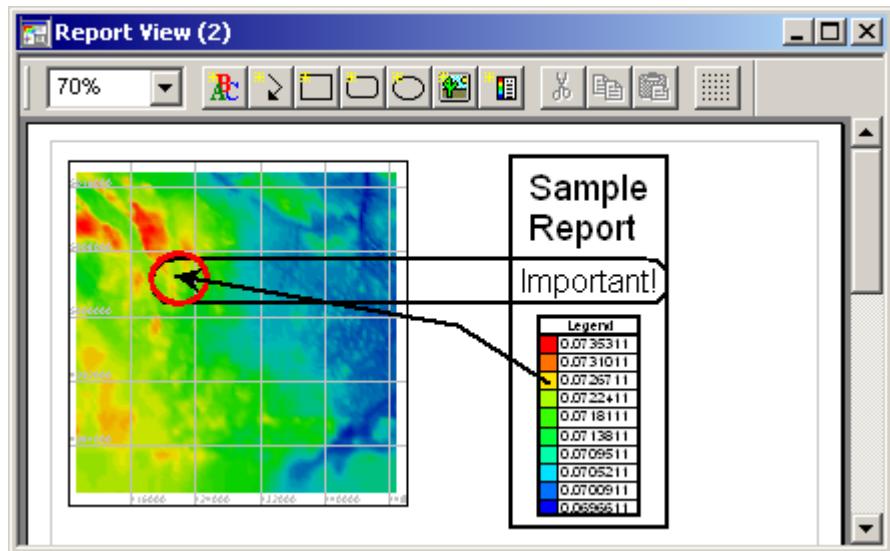
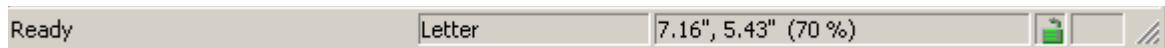


Figure 1.30: The report view can contain other views, as well as decorations

1.5.9.1 The Report View Window Status Bar

The bottom of the EnSim window provides information on the open window.

If the current Medium (See "Report View Window Page Setup Properties", on p. 52) is **Paper**, the status bar for the report view window shows the orientation of the current report, as "Letter", "Legal", or "Custom". The bar also shows the position of the mouse cursor, in inches, measured from the top left corner of the sheet, as well as the current zoom level.



If the current Medium is **Image**, the status bar shows the resolution of the image being created, and the current mouse position in pixels from the top left corner, as well as the current zoom level.



1.5.9.2 The Report View Window Tool Bar

Each Report View window features a built-in tool bar at the top. These buttons allow access to drawing objects, the zoom level of the view, clipboard functions, and the snap-to grid.



- - This menu lets you select the scale at which the report is displayed. The minimum size is 10% (1/10th of actual size) and the maximum is 500%. You can type in a value, or select from the drop-down box.
- - This button allows you to insert a text label on the report. Through the **Properties** dialog, each label can be customized for colour, background border, font, weight, and justification. To change the text of a label, double-click on it twice. See "Labels" under View Decorations, on p. 59, for more information.
- - This button allows you to draw an arrow on the report. Click on the button, then click on the report where you would like the tail of the arrow to be. Click again to connect the two points with a line. If you continue to click, each point will extend the arrow. To stop, click on the button again, or press <Esc>.

To turn an arrow into a line, open up the **Properties** dialog of the arrow (double-click on it to select it) and change the **Head Style** to "none".

- - Click on this button to create a rectangle in the top-left corner of the report. To move the rectangle, click anywhere inside it and drag it to the new position. To resize the rectangle, click and drag on an edge or corner. To change its appearance, right-click on it and open the Properties dialog.
- - This button creates a rounded rectangle in the top-left corner of the report. To move the rounded rectangle, click anywhere inside it and drag it to the new position. To resize the rounded rectangle, click and drag on an edge or corner. To change its appearance, right-click on it and open the Properties dialog.
- - Much like the Rectangle and Rounded Rectangle buttons, this button creates an ellipse in the top-left corner of the report. To move the ellipse, click anywhere inside and drag it to the new position. To reshape or resize the ellipse, click and drag on an edge or corner. To change its appearance, right-click on it and open its Properties dialog.
- - This button opens a dialog box that lets you select a bitmap image to insert into the report. Like the shape objects, you can move or resize the image by clicking and dragging. On the Properties dialog, you can give the image a border, or unlock its aspect ratio to resize its height and width independently.
- - Clicking this button creates a legend in the report. If you have any data items loaded in the Workspace, a dialog box will appear that allows you to select from which object the legend will be created. If you choose an object as the source of the legend, the Colour Scale properties will be used to determine the colours used. If you choose to create a default legend, a simple legend will be inserted.

-  - Clicking this button copies the selected object to the clipboard and removes it from the report.
-  - Clicking this button copies the selected object to the clipboard, but leaves the original behind.
-  - Clicking this button pastes the object in the clipboard into the report. If there is no object available to be pasted, this button will be greyed out.
-  - This button toggles the snap-to grid on and off. See "Report View Window Page Setup Properties", on p. 52 for more details.

1.5.9.3 Manipulating the Report View

The EnSim Report view represents a sheet of paper on which the report is displayed. Instead of manipulating the report itself, you change its appearance and contents by manipulating the objects it contains.

To add a view to a report:

In the Workspace, click on the view that you would like to add. Drag the view down to the title of the report, just as if you were adding a data item to a regular view. Note that the view will now be contained only within the report.

To manipulate a view that has been added to a report:

1. Select the view within the report window by double-clicking on it. The report will be shown with a dashed magenta outline.
2. Double-click on the view again, or select **Activate** from its shortcut menu. An activated view will appear with a raised outline, and can be manipulated as if it were a regular view window. See the section for the type of view you're manipulating for more information.

To change the order of objects in the report:

Objects in the report window are shown with the most recently added objects on top. To reorder them, select the object you would like to move and select **Order** from its shortcut menu.

- Bring to Front: The object will appear on top of any other objects in the report.
- Send to Back: The object will appear beneath any other objects in the report.
- Bring Forward: The object will be raised one position.
- Send Backward: The object will be lowered one position.

To change the border around an object in the report:

Any object in the report view can be displayed with or without a border. To remove or change an object's border, select **Properties** from its shortcut menu.

Note: If you remove the border from a rectangle, rounded rectangle, or ellipse, its outline will disappear. If you haven't given it a background colour, you may have a hard time finding it again.

Note: To change a view's background colour, use the **Display** menu in its **Properties** dialog. You can access that dialog through the Workspace, or by activating the view within the report. See the Display Properties section for the type of view you're changing for more information.

1.5.9.4 Report View Window Page Setup Properties

The page setup properties of the Report View window can be changed in the PageSetup tab of the view's Properties dialog box.

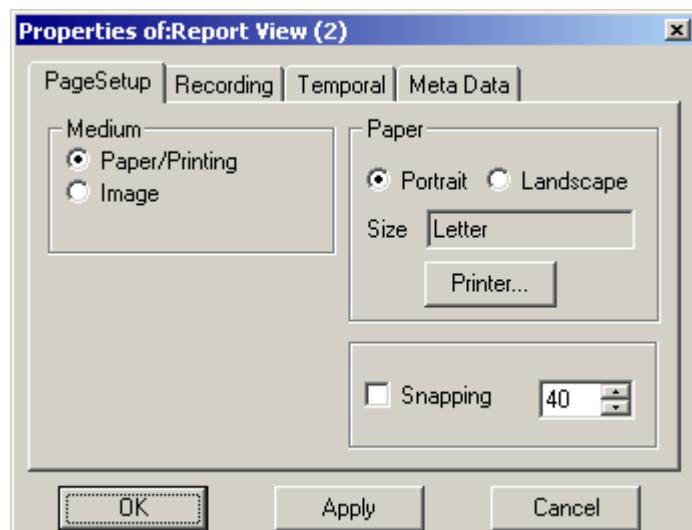


Figure 1.31: This dialog lets you choose the format that the report will take

The page setup properties that can be edited include:

Medium: This can be either **Paper/Printing** or **Image**. If Paper/Printing is active, the Paper properties will be accessible. If Image is active, the Paper properties will be greyed out. You can change the image resolution on the **Recording** tab of the Report's **Properties** dialog.

Paper: This option is only available if the Medium is Paper/Printing. In this area, you can select the orientation and paper size of the report. The orientation can be either Portrait or Landscape. The paper size depends on the capabilities of the active printer, which you can select with the **Printer...** button.

Snapping: This area controls the snap-to grid. If the snap-to grid is active, objects being drawn or resized will automatically snap to the nearest point on the grid. The value next to the checkbox determines the number of points on the grid. The higher the number, the finer the grid will be.

The snap-to grid can also be turned on and off by using the report's shortcut menu or the  button on the report tool bar.

1.5.9.5 Report View Templates

One of the unique abilities of the Report view is its ability to use templates to prepare reports. If you need to prepare multiple reports or pages with a similar appearance, this feature can save a great deal of time.

To create a report template:

1. Create a new report and add objects and views that will be common to all of the reports you need to create. Note that data items will not be saved with the template, although legends based on data items will be saved.
2. Under the Meta Data tab of the report's Properties window, change the report's title to something indicative of its purpose.
3. On the menu bar, select **Window→Save Template....**

To use a report template:

1. On the menu bar, select **Window→Load Template....**
2. From the dialog box, select the template that you would like to use. The template will be loaded as a Report view, with the same titles, views, and objects with which it was saved.
3. Since all reports created from a template have the same titles, it's a good idea to change the titles of the Report and its views on the Meta Data tab of the respective Properties dialogs.

1.5.10 View Decorations

Decoration objects are non-data items that are added to the view window to enhance the presentation of the data. Colour scale legends, compasses, simulation clocks and labels are all decoration objects.

A decoration object is selected by double-clicking on it in the view window. Once selected, the corners of the object are highlighted with magenta squares, and a dashed magenta line surrounds the object.

Once selected, a decoration object can be moved by dragging it to another location in the view window. The decoration object does not move when the data items in the view are moved or resized.

The selected decoration can be resized by clicking and dragging the mouse on one of the magenta squares. The decoration object size is a percentage of the window size; it is resized automatically when the window is resized.

A four-arrow cursor for moving and a two-arrow cursor for resizing differentiate moving and resizing.

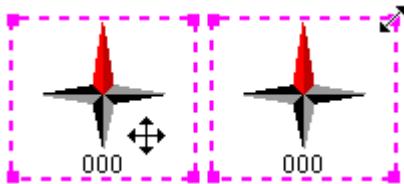


Figure 1.32: The cursor on the left is moving the compass rose; the cursor on the right is resizing it.

The properties of a selected decoration object are accessed by the **Properties** command in the **Edit** menu or in the decoration object's shortcut menu.

Note: To remove a decoration Object, select it and press <Delete> or select **Remove** from the shortcut menu.

1.5.10.1 Legends

There are two types of legends that can be drawn in a view window: The colour scale legend, which is dependent on the selected attribute of an associated data object; and an independent legend, which can be constructed and modified but is not dependent on any one data attribute.

1.5.10.1.1 Colour Scale Legends

A *colour scale legend* is a colour scale drawn in a view window. The colour scale describes one of the data items in the view. The colour scale legend is a decoration object that is accessed through the **Colour Scale** tab of a data item's **Properties** menu. See the "Colour Scale" under Properties of Data Items, on p. 20 for more details.

The colour scale legend can be toggled on or off using the **Show** check box. The options dialog of a colour scale legend is accessed through the **Options...** button.

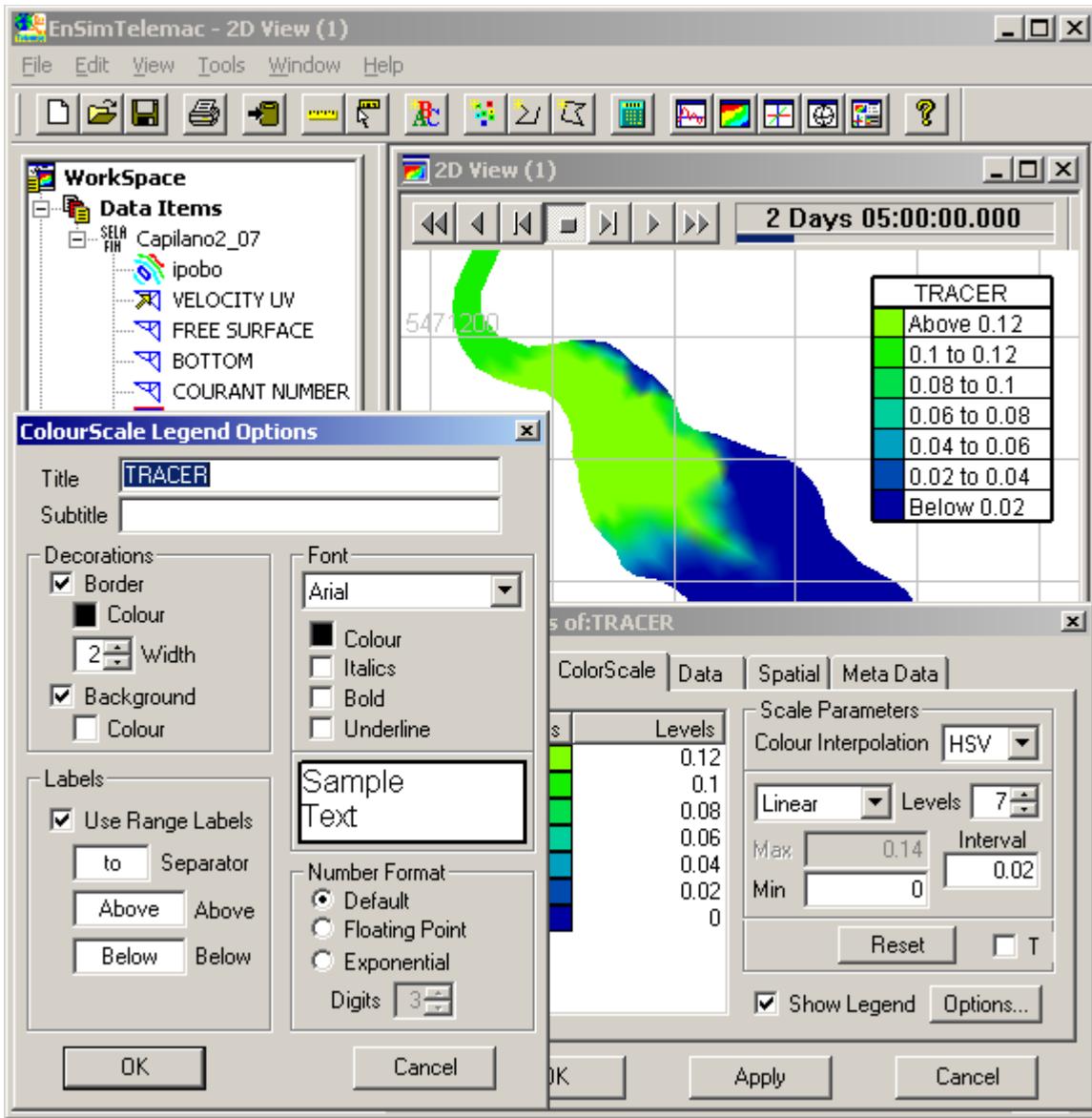


Figure 1.33: The Colour Scale Legend Options dialog of a 2D view. The Properties dialog is shown to its right

The colour scale legend properties that can be edited are as follows:

- **Title, Subtitle:** Both titles are shown at the same font size, with the subtitle appearing below the title. The title text is not wrapped, and the legend width is sized accordingly.
- **Decorations:** This determines the appearance of the colour scale legend **Border** and **Background**. The **Colour** and **Width** of the border and the Colour of the background can be changed.
- **Number Format:** This determines the appearance of the colour level numbers. An option button selects between **Default**, **Floating Point**, or **Exponential** formats. The number of **Digits** after the decimal place can be defined for floating point and exponential formats.
- **Labels:** This determines the appearance of the colour level labels.

- **Use Range Labels:** If this is toggled off, only the number representing the colour level is displayed. When it is toggled on, two numbers of a range representing each colour level are displayed.
- **Separator:** The separator between the two numbers of the range. The default separator is the word "to."
- **Above:** The label of the top colour level, indicating a range greater than a certain number. The default above label is the word "above."
- **Below:** The label of the lowest colour level, indicating a range below a certain number. The default below label is the word "below."
- **Font:** This determines the font of the title, numbers, and labels. The **Font Name** and **Colour** can be edited.

The legend width is determined by the longest of the title, subtitle, or level fields.

1.5.10.1.2 Independent Legends

An *independent legend* may be displayed in any view window. The legend can be both created and edited.

To create an independent legend:

With a view selected, select **View→New Legend**. An empty legend will be drawn within the selected view window and a legend properties dialog will be automatically opened.

To edit an independent legend:

Select the legend within the view by double clicking on it. The properties dialog can be launched by either double clicking again on the selected legend, by selecting **Edit→Properties**, or by selecting **Properties** from the legend's shortcut menu.

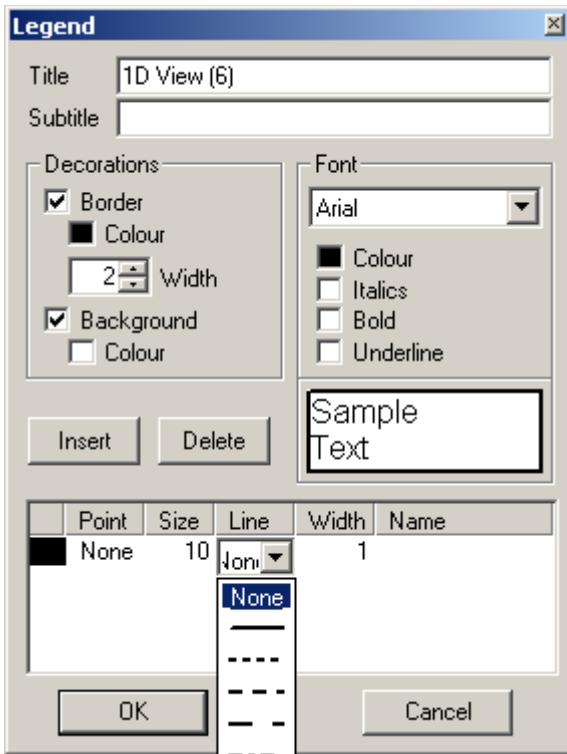


Figure 1.34: The independent legend property dialog

The independent legend properties that can be edited are as follows:

- **Title, Subtitle:** Both titles are shown at the same font size, with the subtitle appearing below the title. The title text is not wrapped, and the legend width is sized accordingly.
- **Decorations:** This determines the appearance of the colour scale legend **Border** and **Background**. The **Colour** and **Width** of the border and the Colour of the background can be changed.
- **Font:** This determines the font of the title, numbers, and labels. The **Font Name** and **Colour** can be edited.
- **Insert:** Clicking on this button will add an item to the legend. Alternatively, clicking below the last item in the list will also add an item.
- **Delete:** Clicking on this button will delete the selected item from the legend.
- **Item Window:** Each item in the legend may be selected and the item's individual properties modified. Colour, point style, line style, line width and name for each individual item can be modified by clicking on the required cell in the item window. The figure above shows a drop down list of potential line style.

Creating a Quick Legend:

For data objects displayed in a 1D view, a Quick Legend is available. The Quick Legend is an independent legend except the item list is populated with all objects currently displayed in the selected 1D view.

To create a Quick Legend, select the 1D view displaying the data objects and then select **Quick Legend** from the 1D view's shortcut menu.

1.5.10.2 The Compass

The *compass* is a decoration object that illustrates the direction of a view with a four-arrow direction indicator. The red arrow indicates North.



Figure 1.35: This compass indicates that the top of the view is Northeast

The compass also shows the bearing or viewing direction in compass degrees. At zero degrees rotation the north arrow points up; at 90 degrees rotation the north arrow points to the left.

Compass properties include a border, background colour, and bearing appearance.

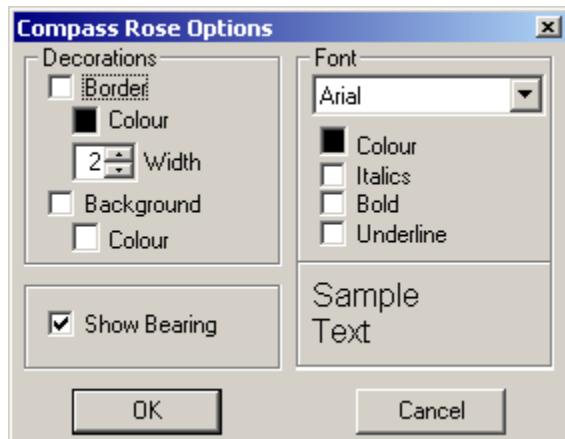


Figure 1.36: These compass properties can be customized

1.5.10.3 The Simulation Clock

The *simulation clock* is a decoration object which illustrates the progression of time during an animation, using a digital counter.

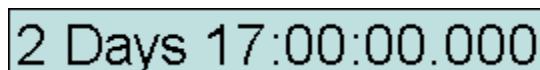


Figure 1.37: This simulation has been running for almost 3 days

Clock properties include number format, border, background colour and font. The properties are accessed in the **Temporal** tab of the view properties. The **Options** button on the **Temporal** tab provides an additional dialog with properties controls.

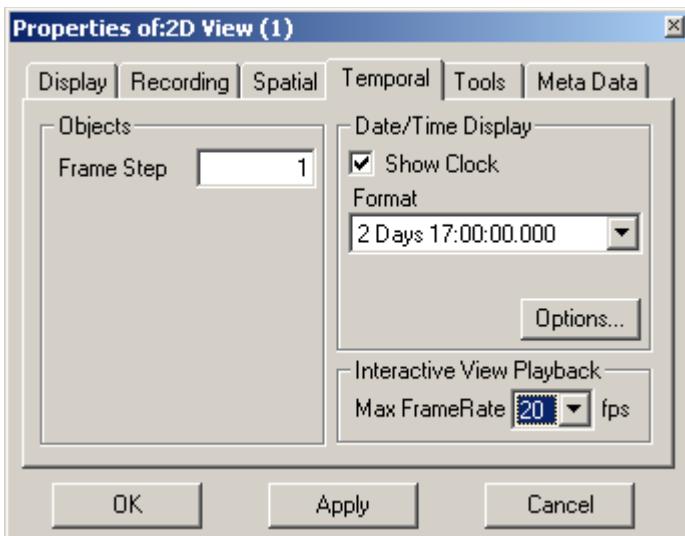


Figure 1.38: The Properties tab for the Simulation Clock

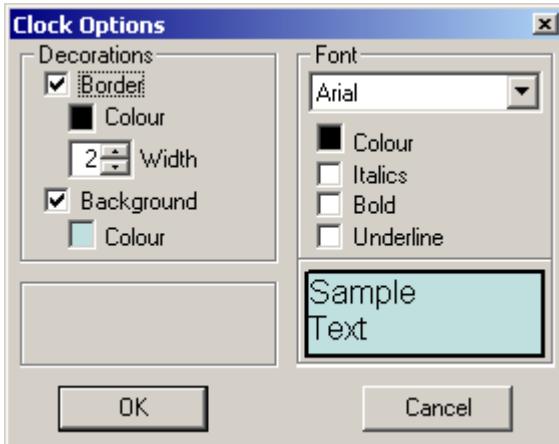


Figure 1.39: This dialog is accessed from the Options button shown in the above image

1.5.10.4 Labels

Labels allow you to display text information within a view window. They can be created and edited.

To create a label:

1. With a view selected, select the create labels button , or **View→New Label**.
2. Click the cursor in the view window. A text window will appear at the cursor location.
3. Type the text that you would like displayed. Note that the label will extend horizontally according to the length of the text, and vertically in response to the <Enter> key.
4. Click the cursor outside the label to finish.

To edit a label:

1. Double-click on a label to select it.
 - To edit the label's text, select **Edit** from the label's shortcut menu.
 - To delete the label, select **Remove** from the shortcut menu, or press <Delete>.
 - To move or resize the label, click on and drag the border as desired. Note that the text within the label will resize automatically.

Label properties can be accessed through the shortcut menu, or by selecting **Edit→Properties**.

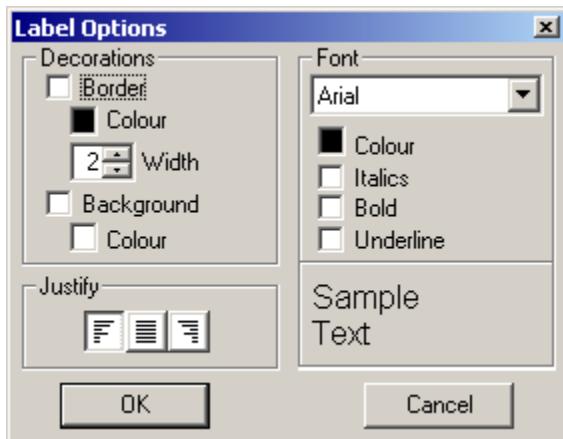


Figure 1.40: The Properties dialog of a Label

- **Font:** Allows the font type, **Colour** and style to be edited. The three font styles available are **Italics**, **Bold**, and **Underline**.
- **Justify:** Select left justified, centred, or right justified.
- **Decorations:** Determines the appearance of the label's **Border** and **Background**. The **Colour** and **Width** of the border, and the **Colour** of the background can also be changed.
- **Sample Text Box:** This shows a sample of text with the current settings.

1.5.11 Displaying Base Maps

Several Base Maps are included with Kenu. These maps can be displayed in a Spherical, 2D or 3D view, and use the LatLong projection.

To Display a Base Map:

1. On the Menu bar, select **File→Base Maps→(Scale)→(Map Name)**

The selected Base Map will appear in the current View, if the view can display the map. Appropriate views include the 2D, 3D, and Spherical views. If the view cannot display the Base Map, the map will be loaded in the WorkSpace, but not displayed. You can display the map by dragging it to a 2D, 3D, or Spherical view.

The Base Maps are included in several scales:

- **1:20,000,000** - The **World** map shows national borders for the entire world.
- **1:7,500,000** - The **Canada** map shows provincial and territorial borders within Canada.
 - The **Rivers and Lakes** map shows major rivers and lakes within Canada.
 - The **Roads** map shows major roads within Canada.
 - The **Cities** map shows medium-to-large settlements within Canada. The names can be shown or hidden by checking or unchecking the **Show Node Labels** box in the Fonts section of the **Display** tab of the **Properties** menu. The Cities_7.5m map contains data, including NTS50 values and populations (as of 1991) for 497 settlements. The additional data can be viewed by right-clicking on the data object and selecting **Show Attribute Table** from the shortcut menu.
- **1:1,000,000** - The **Rivers and Lakes** map shows most rivers and lakes within Canada.
 - The **Roads** map shows most roads within Canada.
 - The **Cities** map shows most settlements within Canada. Like the Cities_7.5m map, the settlement names can be shown or hidden. The Cities_1m map contains location data for 3540 settlements.
- **1:50,000** - The **Map Sheets** map shows a grid covering all land within Canada. This grid corresponds to the NTS map sheet boundaries published by Natural Resources Canada.
 - The **Map Sheet Names** map contains the names for each of the grids outlined in the Map Sheets map. The two maps can be used together to quickly determine which maps might be needed from websites such as GeoGratis or GeoBase.

1.5.12 Animation

Time-varying data can be animated by toggling on the **Animate** check box in the **Display** tab of the data item's **Properties** dialog box or through the object's shortcut menu. Data that does not vary over time will not have this checkbox.

When the data item is toggled on to animate, the animation tool bar appears. The animation tool bar can also be removed or reinstated by selecting the view window and selecting **View→Animation Bar**.

The position of the animation tool bar can be moved. The default position of the animation bar is docked at the top of the view window. Note that each view window has its own animation bar. That is, the animation bar from view window 1 is used to animate the data in view window 1; it is not used to animate the data in view window 2

The animation tool bar looks like this:



The tool bar buttons, from left to right are:

jump to the first frame

-  play in reverse
-  step backwards one frame
-  stop
-  step forwards one frame
-  play
-  jump to the last frame

The animation tool bar has a text box in which the frame, step, or time counter can be displayed. The counter can be switched from frame, step, or time counter by clicking on the time.

Below the text box is an animation progress meter. The animation scroll feature displays the progress of the animation. The progress meter can be moved to a specific point within the animation by clicking the left mouse button on the bar and dragging the bar to the desired position.

To save an animation as an .avi file, see "Recording" under Saving and Copying Images, on p. 65.

1.5.13 Flight Paths

A flight path consists of a series of viewpoints within a 3D or Spherical view. The flight path can be viewed and recorded as an animation, allowing you to emphasize certain locations or perspectives within the view, or to provide a visual summary of a data item.

To create a new flight path:

1. On the menu bar, select **View→New Flight Path...**. A new flight path object, called "new Flight Path," will appear in the Workspace, and its Properties window will open. Click and drag the flight path into the appropriate view. Note that flight paths can only be added to 3D or Spherical views.
2. Within the view window, move the view into the perspective and location from which you would like to start the flight path.
3. Click **Add**.
4. Move the viewpoint within the view window to the next control point along the flight path. Control points should be locations where the flight path curves. Placing multiple control points along a straight line usually has the same effect as placing points at the beginning and end of that line.

As you add control points, EnSim calculates the cubic spline between each, providing a smoothly curving path that intersects each point.

1.5.13.1 Flight Path Properties

The Properties dialog of a flight path can be accessed through its shortcut menu, or by double-clicking on its name in the WorkSpace.

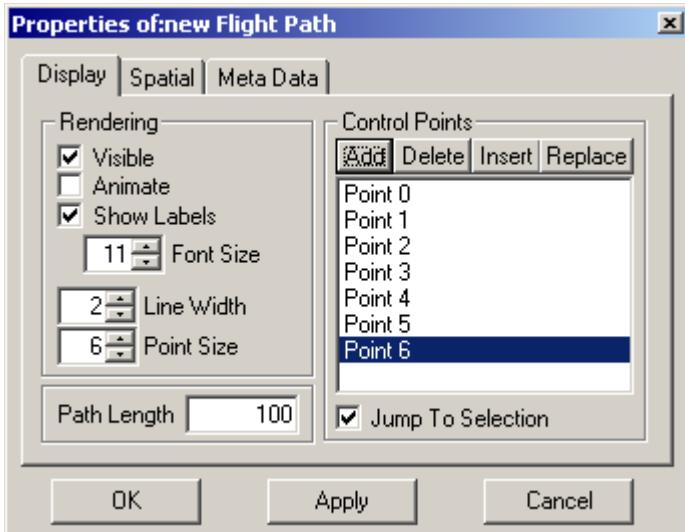


Figure 1.41: This flight path has seven control points

Rendering: This area controls the appearance of the flight path. The **Visible** and **Animate** checkboxes may also be toggled through the flight path's shortcut menu.

Path Length: This value determines the number of steps used to travel along the flight path. The greater the number entered, the more slowly the path will be animated.

Tip: To slow down the animation while keeping the movement smooth, increase both the Path Length and the Frame Rate. The Frame Rate can be found on the View's Temporal Properties tab.

Control Points: This area lists each control point used to construct the flight path. You can examine a control point by checking **Jump To Selection** and clicking on the point name.

Add: This button adds the current perspective to the end of the flight path.

Delete: This button removes the currently selected control point.

Insert: This button adds the current perspective to the flight path above the currently selected control point.

Replace: This button changes the currently selected control point to the current perspective.

1.5.14 Synchronizing Two Views

The ability to synchronize two views is a very useful feature in EnSim. This feature allows you to observe the value of a specific point over time within a multidimensional View. An example of this might be the synchronization of an animated data item in a 2D View (Source View) with a time series extracted from a selected point of the animated data item (Result View). Currently, 1D Views can receive synchronization from 2D, 3D or Spherical Views.

To synchronize Views:

1. Load a temporal data item into the WorkSpace.
2. Select a point on the data item and extract the time series from the selected point. See "Extracting Time Series" under Extracting Data, on p. 96 for more details.
3. Drag the extracted time series into a 1D View (the Result View).
4. Ensure that the 1D view is the current view. Select the **View→Select Sync. View...** option. The following dialog will appear:

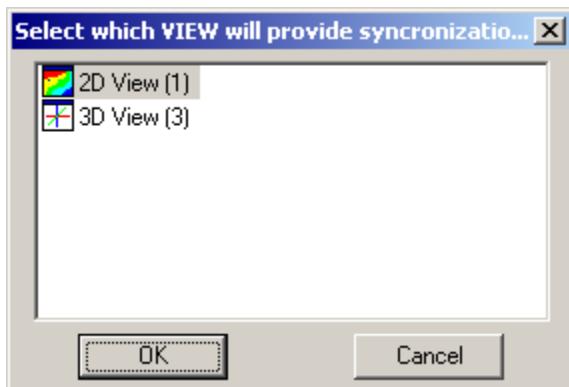


Figure 1.42: This dialog is used to synchronize two views

5. Select the View that will provide the synchronization (Source View) and press the **OK** button.
6. Animate an object in the Source View. Refer to "Animation" under Views, on p. 61 for more details on animating a data item. A vertical red line will appear in the Result View. While the Source View is being animated, the vertical red line in the Result View will be redrawn at the same temporal location as the current Source View.

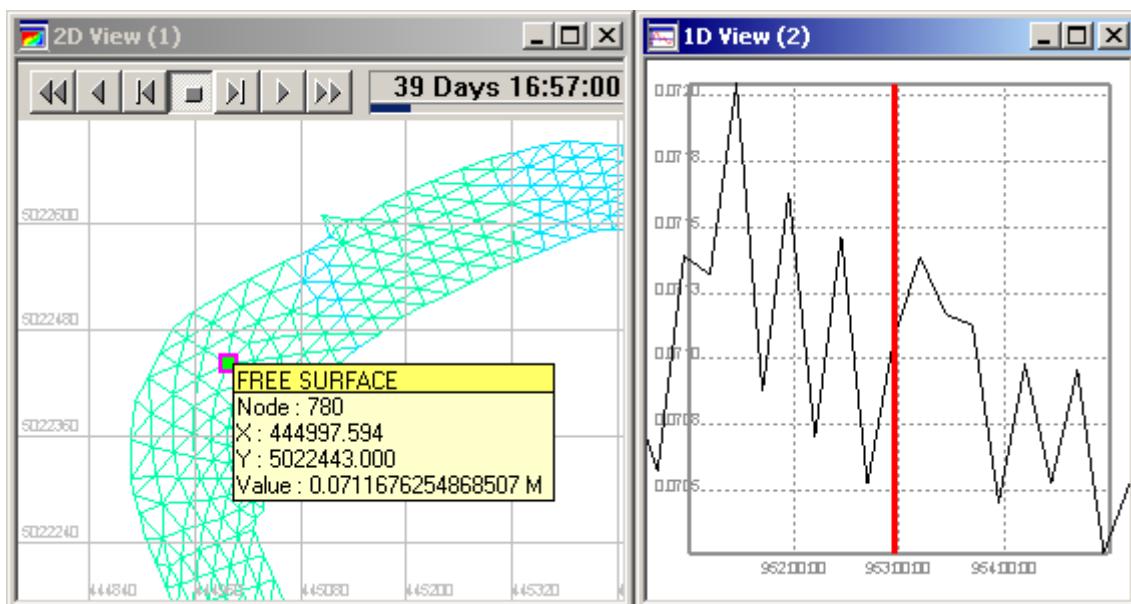


Figure 1.43: These two views have been synchronized

1.5.15 Saving and Copying Images

Images from a View can be copied or saved for use in other Windows applications. A "snapshot" of the current, static display in a view can be copied for use in a word processing application, to add the image to a report; it can be copied to presentation software; or copied to a photo editor to save the image for repeated use. Static images can be printed to produce a quick paper copy of the View window display. Time-varying, or dynamic, images can be saved by recording the display data as a movie or animation (in *.avi format).

1.5.15.1 Recording

A movie can be created showing animated data in the view window. Movies created in EnSim are in *.avi format.

To create a movie:

1. In the View window **Properties** dialog, select the **Recording** tab.

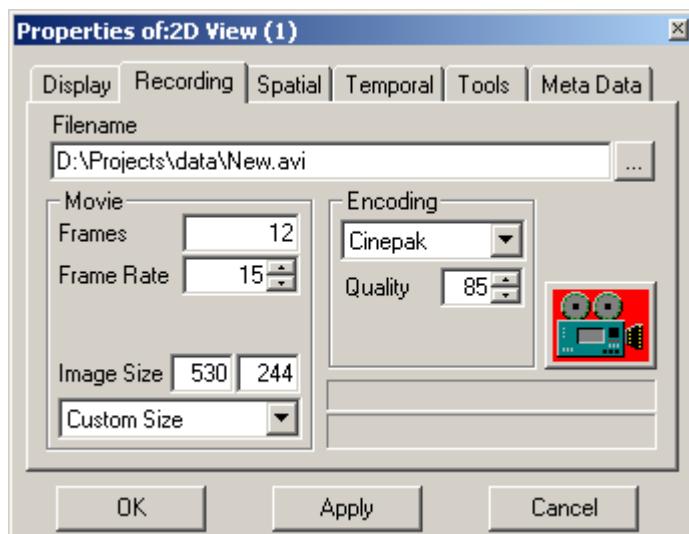


Figure 1.44: The Recording tab is found in the Properties dialog of the view

2. Provide a **Filename** for the recording. Use the button to browse.
3. In the **Movie** box, select the number of **Frames** to be recorded. The movie will begin at the current frame in the view window. For example, if you want to avoid recording the first 10 frames of an animation, begin record when the counter is on frame 11. If more frames are indicated than remain in the animation, the final frame will be repeated.
4. The **Frame Rate** is the number of frames per second of playback. A frame rate of 15 is generally good.
5. **Image Size** refers to the size of the view window for the recording. Choose a standard window size from the menu, or create a custom size by entering the width and height, respectively, in pixels. Note that the NTSC and PAL standards are provided.
6. **Encoding** is the type of compression used in creating the avi record. Different types of encoding will produce different quality recordings. The standard Windows encoding

methods are provided in the menu. **Cinepak** is the most widely-used method. Other choices are available under the **Advanced** option. With the **Advanced** option, a second dialog, **Video Compression** will appear once the movie when this dialog is closed.

7. **Quality** allows you to adjust the quality of the recording, and as a result, the size of the file produced. File size and quality are not linearly related. That is, decreasing the quality to 50% will not reduce the file size by half, but will produce a smaller file. 85% is the recommended quality level.



Figure 1.45: This button is used to start recording

8. When you are ready to begin record, click the **Record** button. If you selected **Advanced** encoding the **Video Compression** dialog will appear next. While recording, the view window may not update to show the animation in progress.

1.5.15.2 Copying to the Clipboard

A bitmap image of a view may be *copied to the clipboard* and pasted into other applications. NOTE: The image stored in the clipboard will have twice the number of pixels in X and Y as the source view window.

To copy the image of a view window to the clipboard:

1. Arrange the view as you wish to record the image. Ensure that the view window is the currently selected view.
2. Click on the **Copy to Clipboard** button  in the tool bar, or select **View→Copy to Clipboard**.

The image may then be pasted by using <Ctrl+v> or selecting **File→Paste**.

1.5.15.3 Printing

Selecting the **Print** command from the **File** menu will send the image in the currently selected view window to the printer destination designated in the **Print** dialog.

A title block will be given to the printed image, displaying the view window title and the date and time at which the view was printed. To change the title of the printed image open the **Properties** dialog of the View window. Choose the **Meta Data** tab and edit the **Title** and **Subtitle** fields. The **Subtitle** will be placed under the **Title** on the printed image.

1.5.16 Troubleshooting in Views

If the object is not displayed after it is dropped into a view, try the following:

- Choose **Default View** from the View's shortcut menu to centre the view on the object.
- If there is more than one object displayed in the view, make sure that their coordinates are compatible.

- In 1D, Polar, or 2D views, try zooming out.
- In 3D views, open the **Properties** dialog of the data item and choose the **Display** tab. Set the **Scale** and **Shift** parameters of all objects to 1.

1.6 TOOLS

Tools in EnSim are functions that allow you to manipulate data, retrieve information, and create data items. They help you to analyze data and to understand model output results.

These tools include:

- Creating New Data Items
- Editing Data Items
- Probing Data
- Extracting Data
- Time Series-related Tools
- Creating Vector Fields
- Mapping Objects
- Calculators

Following the descriptions of the tools is the **How To - Hints and Tricks** section, which describes how these can be used for completing tasks that may not be readily apparent.

1.6.1 Creating New Data Items

1.6.1.1 Drawing Points

EnSim has the ability to define and edit new Point Sets. These points may be used for a number of uses, or they may be used in conjunction with other tools, such as Map Objects. Points can also be digitized from a GeoTIFF. Refer to the section Digitizing From an Imported Image for further details. Points have a location as well as values or attributes.

Points can be saved in *.xyz, *.shp, and *.mif formats.

To create a point set:

1. Open or select a 2D view.
2. Select the  button, or select **File**→**New**→**Points...**. The button will appear depressed: . In the **File**→**New** menu, **Points...** will appear with a checkmark.
3. Click in the 2D View at a specific location to create the first point of the set. Each click will create a new point.
4. To end the point set, reselect the  button, or press <Esc>. The Points button will appear raised. Alternatively, reselect **File**→**New**→**Points...**.
5. A dialog will appear. Enter a name for the new point set. If no name is entered, it will be named "newPointSet" by default. Units may also be entered as well.

Once the Point set has been created, the x- and y-coordinates, as well as the value (or z-coordinate) can be adjusted. Select the point and click on **Edit** on the shortcut menu.

Hint: Lock the view while drawing points, using the **Display** tab of the View's **Properties** dialog. Otherwise, a mouse click may pan the view, instead of creating a point.

1.6.1.2 Drawing Lines and Closed Polylines

EnSim has the ability to define new Line Sets. These lines may be used be used in conjunction with other tools, such as Map Objects. Points and Lines have a location as well as values or attributes.

Lines or closed lines may be saved in *.i2s, *.i3s, *.xyz, *.mif, or *.shp format.

To create a line or polyline:

1. Open or select a 2D view.
2. Select the  button, or **File**→**New**→**Open Line...**. The button will appear depressed: . In the **File**→**New** menu, **Open Line...** will appear with a checkmark.
3. Click within the 2D view to create the first point of the line. Each click will create a new point, with a line connecting it to the previous point.
4. To end the line, reselect the  button, or press <Esc>. The Line button will appear raised. Alternatively, reselect **File**→**New**→**Open Line...**.
5. A dialog will appear, asking you to enter a name, a value, and units for the new line.

Hint: Lock the view while drawing a line, using the **Display** tab of the View's **Properties** dialog. Otherwise, a mouse click may pan the view, instead of extending the Line Set.

To create a closed line or polygon:

1. Open or select a 2D view.
2. Select the  button, or **File**→**New**→**Closed Line...**. The button will appear depressed: . In the **File**→**New** menu, **Closed Line...** will appear with a checkmark.
3. Click within the 2D view to create the first point of the polygon. Each click will create a new point, with lines connecting it to the previous point and to the first point, producing a closed polygon.
4. To end the closed line, reselect the  button, or press <Esc>. The Closed Line button will appear raised. Alternatively, reselect **File**→**New**→**Closed Line...**.
5. A dialog will appear, asking you to enter a name, a value, and units for the new closed line.

Hint: Lock the view while drawing the line, using the **Display** tab of the View's **Properties** dialog. Otherwise, a mouse click may pan the view instead of extending the Closed Line.

1.6.1.3 Creating a New Regular Grid

To create a new Regular Grid:

1. Create a new grid object by selecting **File**→**New**→**Regular Grid....** The grid object that appears in the WorkSpace will be empty.
2. Open the **Properties** dialog of the grid object and choose the **GridGen** tab. The fields will be empty.

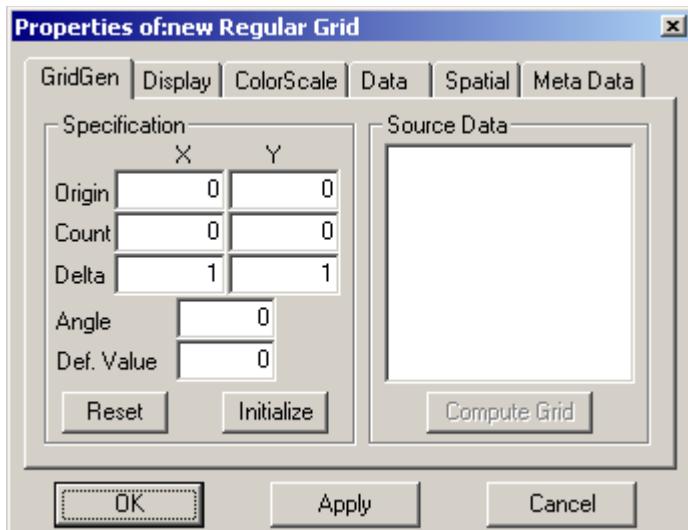


Figure 1.46: The GridGen tab of a Rectangular Grid, with sample data

3. If source data is being used, drag the source object(s) into the empty grid object in the WorkSpace. The source data can be one or more point set (*.xyz), parcel set (*.pcl), line set (*.i2s), triangular grid (*.t3s) or rectangular grid (*.r2s) objects. Once they are added to the grid object, they will appear as children of the grid object in the WorkSpace. The source data will appear in the **Source Data** box of the open Properties dialog. EnSim calculates the Origin and Delta based on the default Count of 100.
4. Click on the **Initialize** button to create a grid with the default parameters. The icon of the grid object will change, indicating that the new grid object now contains data.
5. Drag the grid object and the source data item(s) into a 2D view. Edit the **Specification** options of the grid until the grid has the desired location, orientation and resolution.

The **Specification** options are as follows:

- **Origin:** The location of the node in the Southwest (bottom left-hand) corner of the grid.
- **Count:** The number of nodes (not elements) in the x and y directions.
- **Delta:** The interval between nodes in the x and y directions.
- **Angle:** Rotates the grid at the angle specified in the clockwise direction. The origin is the point of rotation, and is thus not moved.
- **Default Value:** Provides a value to be given to each node of the grid. For example, this value might represent the elevation at each node in the grid.

The **Reset** button will return these specifications to their default values and update the grid displayed in the view.

6. To view the changes to the grid based on the edited specifications, click on the **Initialize** button. The grid displayed in the view will be updated.
7. When the grid is satisfactory, click on the **Compute Grid** button. EnSim will initialize the grid and then interpolate onto the grid the node values from a triangulation of all source data. Isolines will be divided into sections based on the grid size, and the vertices of the sectioned line will be used in the interpolation.

Note: The Properties dialog of the grid contains all the tabs of a viewable object in addition to the **GridGen** tab. See "Properties of Data Items" under Data Items, on p. 17, for more details.

1.6.1.4 Creating a New Triangular Mesh

This section explains how to create a triangular mesh from an existing set of points.

To create a new Triangular Mesh from existing data:

1. Open the source data file. This might be a point set (*.xyz), parcel set (*.pcl), line set (*.i2s), triangular mesh (*.t3s) or regular grid (*.r2s).
2. Create a new triangulation grid object by selecting **File→New→Triangulation....** The triangulation object that appears in the WorkSpace will be empty. The **Properties** dialog of the triangulation object will open automatically.
3. In the WorkSpace, drag the source data into the triangulation object. The data points will be shown as children of the triangulation object. More than one file or object can be used as source data for the triangulation. The **Source Data** for the triangulation will be listed in the **Properties** dialog.

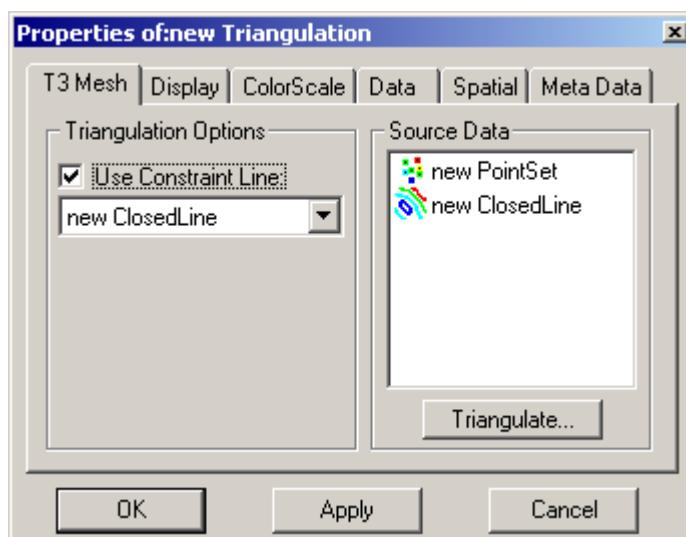


Figure 1.47: The T3 Mesh tab of a new Triangular Mesh, with "new PointSet" as source data

You can select a closed polygon as a constraint line which will be used as the outer boundary of the triangulation.

4. Select the **Triangulate...** button in the T3 Mesh tab. This will create a triangular mesh based on the source data points, according to the specifications described in the previous step.

Note: The Properties dialog contains all the tabs of a viewable object in addition to the **T3 Mesh** tab. See "Properties of Data Items" under Data Items, on p. 17, for more details.

1.6.1.5 Creating a New Table Object

This section explains how to create a new table object and populate it with time series data. To extract an attribute table from an existing data object, see "Extracting an Attribute Table", on p. 100.

To create a new Table Object from existing data:

1. Create a new grid object by selecting **File**→**New**→**Table Object....** The table object that appears in the WorkSpace will be empty.

If the **Properties** dialog of the table object is opened and the **Data** tab is selected, the attribute list will be empty.

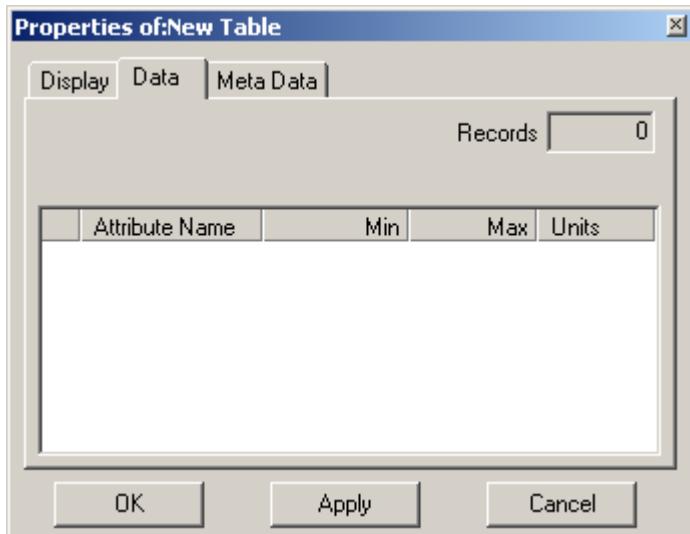


Figure 1.48: The Data tab of an empty Table Object

2. To build the table object, drag the source object(s) into the empty table object in the WorkSpace. The source data must be a time series object. Once they are added to the table object, they will appear as attributes on the Data tab.

Note: The first time series added to the table object will define the start time, the deltaT and the item count. To successfully add subsequent time series to the table object, they must have the same temporal attributes and item count as the first.

1.6.2 Selecting Data Items

In order to edit, probe or extract detailed data associated with individual points, lines, nodes, cells, etc. the EnSim interface provides tools to select these individual items. In general the user simply double clicks on a data object in a view and the item under the cursor is highlighted in magenta indicating its selected state. The generic term "item" is used on the interface since its definition depends on the current data object (ie. point, line, node, etc.).

If the id of the item is known (ie. node number in a mesh) the **Edit→Select→Single Item...** menu command allows the user to enter the numeric item id to be selected.

Some data objects such as r2c grids support a "multi-select" feature. For these objects, holding down the <ctrl> key, while double clicking in a view, toggles the selection state of the item under the cursor.

Items belonging to data objects supporting "multi-select" may also be selected geometrically.

The **Edit→Select→Items Along Line...** menu command presents the user with a list of open and closed lines existing in the WorkSpace. Once a line is chosen, all items intersected by this line are selected.

The **Edit→Select→Items In Polygon..** menu command presents the user with a list of closed lines (polygons) existing in the WorkSpace. Once a line is chosen, all items whose centers fall within the closed line are selected.

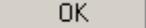
1.6.3 Editing Data Items

This section covers the editing of data items, including regular grids, triangular meshes, line sets, point sets, parcel sets, networks, and time series. Components of foreign (but supported) files, such as MapInfo Interchange or ArcView Shapefile, cannot be edited directly. However, they may be saved in a native EnSim format, which may then be edited.

Editing data items in EnSim consists of altering the value of the current data attribute of a displayed object. In some cases, the coordinates of nodes or points of the object can be edited as well. For information on editing a Time Series, "Editing Time Series" under TimeSeries Tools, on p. 104.

1.6.3.1 Editing Attributes

To edit an attribute of a data object:

1. Select the component to be edited (i.e. the node, line segment, point, etc.), and select the **Edit...** command from the shortcut menu. A dialog box will appear, which differs depending on the type of object and the number of data attributes it has.
2. To edit an attribute value, click on the value to highlight it. Enter the new value and press .

- For a triangular mesh or xyz data with one data attribute, both the data attribute value and the x- and y-coordinates at the node or point may be edited. The **Edit** dialog that appears is nearly identical for the two objects. The dialog for a T3 mesh is shown below. The dialog for an xyz point set would ask you to specify the new values of the point, instead of the Node.

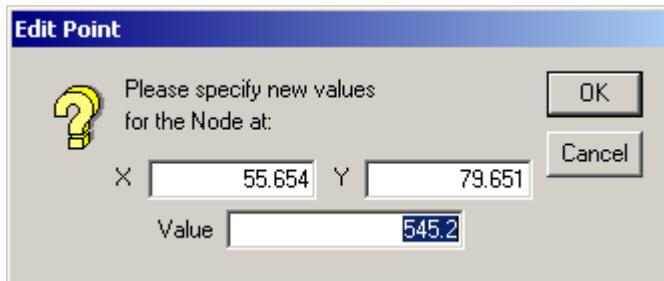


Figure 1.49: This edit dialog is for a triangular mesh data item

- For a regular grid, the position of the node is fixed and only the value of the current data attribute at the node may be edited. The dialog for a regular grid object with only one data attribute is as follows:

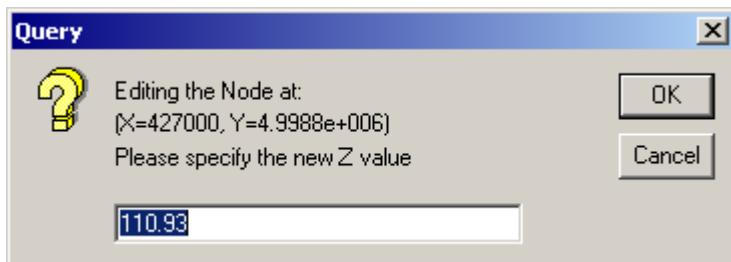


Figure 1.50: This edit dialog is for a rectangular grid data item

- The dialog for a line set object with only one data attribute is similar to that of the regular grid, since only the value of the selected line can be edited. Like the query dialog for the regular grid (see Figure 1.41 on p. 63), it has only one field.
- Objects with multiple attributes share a similar dialog window for editing. An example is shown below.

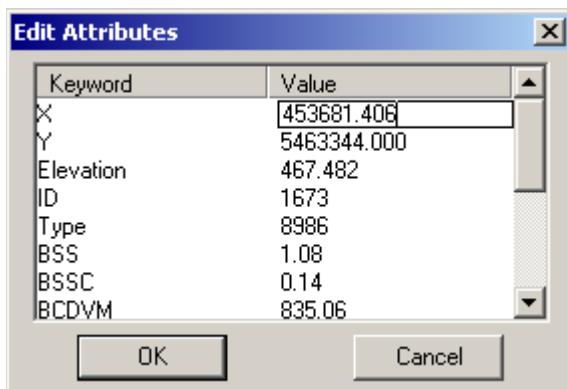


Figure 1.51: This dialog box is used for objects with multiple attributes

The column at the left of the dialog lists the names of all the attributes the object possesses. The column at the right lists the values of the attributes. All attributes can be edited from this dialog, not just the current attribute or the one being displayed in a view. Multi-data attribute objects in EnSim include parcel sets, line sets, r2c objects, and networks.

You can select **Edit**→**T3 Mesh**→**Undo Edit** to reverse the previous edit. **Undo Edit** can be used repeatedly.

1.6.3.2 Editing Points

To edit a point within a point set:

1. Select the point or node to be edited, and select **Edit...** from the shortcut menu or **Edit**→**LineSet**→**Edit...** from the menu bar. The dialog box that appears is similar to the dialog box that appears when a triangular mesh or an xyz data item has been selected. An example is shown below.

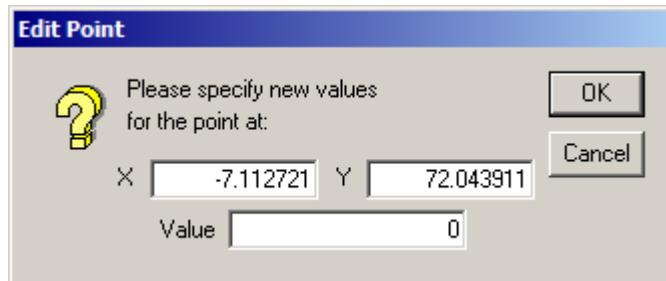


Figure 1.52: This dialog box is used to edit points

2. The X- and Y- coordinates and the value of the selected point can be changed. Once the new values have been entered, click **OK**.

To add a point:

1. Select the Point Set to which you'd like to add points in the WorkSpace, or select a point from the Point Set in the View.
2. From the shortcut menu, select **Add Points...**. The **New PointSet** button on the toolbar will be selected and the cursor will change accordingly.
3. Within the View, draw the new points. See "Drawing Points", on p. 68 for more information.
4. When you're finished, hit **<Esc>** or click  on the tool bar.

To delete a point:

- Select the point or node to be deleted and select **Delete Selected Point** from the shortcut menu, or **Edit**→**LineSet**→**Delete Selected Point** from the menu bar.

1.6.3.3 Editing Lines

To edit the value of a line:

1. Within the View, double-click on the segment of the line set that you'd like to edit. The segment will turn magenta to indicate that it has been selected.
2. Select **Edit→Line Set→Edit...** from the menu bar, or **Edit...** from the shortcut menu.



Figure 1.53: The value of the selected line is being changed to 22

3. Enter the the value for the line and click **OK**.

To edit the location of a point within a line set:

1. Select the point or node to be edited, and select **Edit Selected Point...** from the shortcut menu or **Edit→LineSet→Edit Selected Point...** from the menu bar. This edit option will only appear when a polyline has been selected. The dialog box that appears is similar to the dialog box that appears when a triangular mesh or an xyz data item has been selected. The difference with this box is that the attribute cannot be changed. An example is shown below.

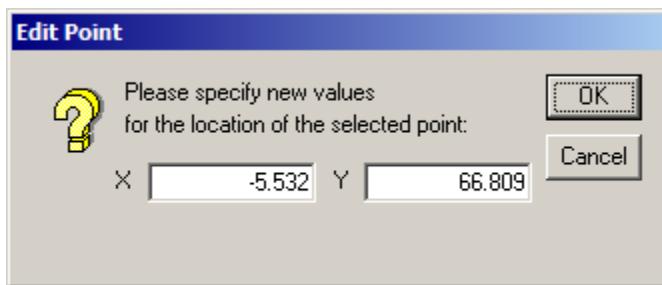


Figure 1.54: This dialog box is used to edit points within line sets

2. The X and Y coordinates at the selected point on the 2D or 3D Line can be changed. Once the coordinates have been changed, press **OK**.

1.6.3.3.1 Transferring Lines within Line Sets

To transfer a line from one Line Set to another:

1. Within the View, select the line segment that you'd like to transfer.
2. Select **Edit→Line Set→Cut** from the menu bar, or **Cut** from the shortcut menu.

The line segment has now been removed from its Line Set and stored in memory. You can now move it to a different Line Set by pasting or appending.

3. In the WorkSpace, select the Line Set to which you'd like to add the line segment.

4. Select **Edit→Line Set→Paste** from the menu bar, or **Paste** from the shortcut menu.

The line segment has now been added to the new Line Set. To add the line segment to more Line Sets, select each of them and select **Paste** again.

To duplicate a line in two or more Line Sets:

1. Within the View, select the line segment that you'd like to duplicate.
2. Select **Edit→Line Set→Copy** from the menu bar, or **Copy** from the shortcut menu.

The line segment has now been stored in memory, but also remains in its original Line Set.

3. In the WorkSpace, select the Line Set to which you'd like to add the line segment.

Note: You can also select the Line Set from which you copied the line segment in order to create multiple copies of the segment within the Set. These copies can then be assigned different values, for example.

4. Select **Edit→Line Set→Paste** from the menu bar, or **Paste** from the shortcut menu.

1.6.3.3.2 Editing Line Segments

To append a line segment to another line segment:

1. Within the View, select the line segment that you'd like to append and **Cut** or **Copy** the segment as described above.
2. Select the line segment to which you'd like to add the first segment. This can be part of the same Line Set or a different Line Set, in the same View or a different View.
3. From the menu bar, select **Edit→Line Set→Append**, or select **Append** from the shortcut menu.

The line segments will be joined at their closest endpoints.

To turn a Closed Line into an Open Line:

1. Within the View, select the Closed Line that you'd like to open.
 - You can select the Closed Line itself by double-clicking on the line between points, or you can select a particular point by double-clicking on it.
 - If you've selected the line, only the first point will be highlighted; if you've selected a particular point, both it and the first point will be highlighted.
2. Select **Edit→Line Set→Open Selected Line** from the menu bar or **Open Closed Line** from the shortcut menu to turn the closed line into an open line.
 - If you've selected the line or the first point on the line, the line segment connecting the first and last points on the line will be removed.
 - If you've selected a point on the line other than the first, the line segment connecting the point with the previous point will be removed.

Note: You can undo the Opening by selecting the Open Line and selecting **Close Selected Line** from the shortcut menu.

To divide a Closed or Open Line into two Open Lines:

1. Within the View, select the point on the Line where you would like the separation to take place.
2. From the menu bar, select **Edit→Line Set→Split Selected Line**, or select **Split Selected Line** from the shortcut menu.
 - If you selected a point on an Open Line, the Line will be divided into two lines whose end points are in the same location as the selected point. No line segments are removed.
 - If you selected a point on a Closed Line, the Line will be divided into two lines whose end points are at the same location as the selected point and at the first point in the line. No line segments are removed.

Note: You can undo the split by selecting either Open Line, selecting **Cut** from the shortcut menu, selecting the other Open Line, and selecting **Append** from the shortcut menu.

1.6.3.3.3 Adding Line Segments

To add an Open Line to a Line Set:

1. In the WorkSpace or the View, select the Line Set to which you'd like to add an Open Line.
2. Select **Edit→Line Set→Add Open Line** from the menu bar or **Add Open Line** from the shortcut menu.

The **New Open Line** button on the tool bar will be selected and the mouse pointer will change accordingly. See "Drawing Lines and Closed Polyline", on p. 69 for more information on drawing an OpenLine.

3. When you're finished drawing the Open Line, click  on the tool bar, or hit <Esc>. You'll be prompted to supply a value and units for the new Open Line.

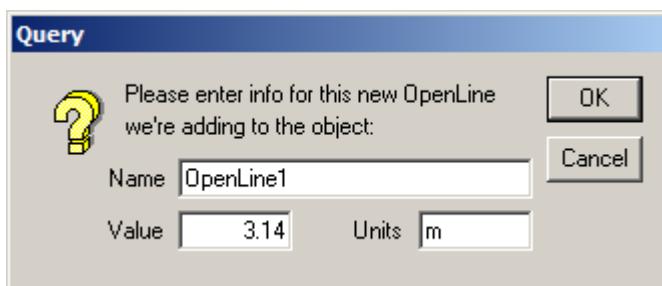
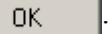


Figure 1.55: This Open Line is being added to the Line Set OpenLine1

4. Click .

To add a Closed Line to a Line Set:

1. In the WorkSpace or the View, select the Line Set to which you'd like to add a Closed Line.

2. Select **Edit**→**Line Set**→**Add Closed Line** from the menu bar or **Add Closed Line** from the shortcut menu.

The **New Closed Line** button on the tool bar will be selected and the mouse pointer will change accordingly. See "Drawing Lines and Closed Polylines", on p. 69 for more information on drawing a Closed Line.

3. When you're finished drawing the Closed Line, click  on the tool bar, or hit <Esc>. You'll be prompted to supply a value and units for the new Closed Line.

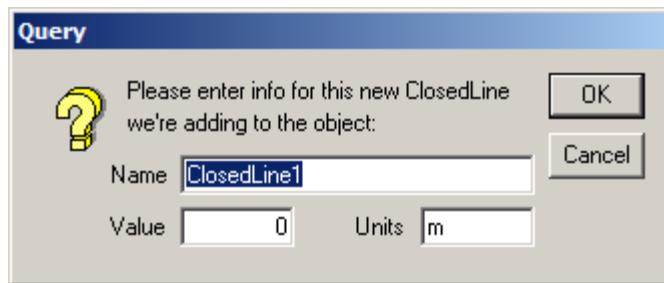


Figure 1.56: This Closed Line is being added to the Line Set ClosedLine1

4. Click .

1.6.3.4 Editing T3 Meshes

The T3 Mesh editing toolbar contains shortcuts to several functions that can greatly simplify the creation and editing of triangular meshes.

To show the T3 Mesh Toolbar, select **View**→**T3 Mesh Toolbar** from the menu bar.

-  - **Edit**→**T3 Mesh**→**Swap Edge**: This tool is available when an internal edge of a mesh is selected. When used, it rotates the selected edge so that it connects the two nodes adjacent to its previous nodes.
-  - **Edit**→**T3 Mesh**→**Split Edge**: This tool is available when an internal edge of a mesh is selected. It places a node at the center of the selected edge and connects the node to its neighbours, creating two new elements.
-  - **Edit**→**T3 Mesh**→**Delete Edge**: This tool is available when an internal edge of a mesh is selected. It removes and collapses the selected edge, merging its source nodes, and effectively deleting the two elements. The number of the new node is that of the lower-numbered source node.
-  - **Edit**→**T3 Mesh**→**Delete Element(s)**: This tool is available when any element or internal edge of a mesh is selected. It removes the selected element or elements, creating a hole in the mesh.
-  - **Edit**→**T3 Mesh**→**Split Element in 2**: This tool is available when an element at the edge of the mesh is selected. It places a node at the center of the exterior edge and connects it to its opposite neighbour.

-  - **Edit**→**T3 Mesh**→**Split Element in 3**: This tool is available when any element is selected. It places a node at the center of the element and connects it to all three of its neighbours, replacing the element with three smaller elements.
-  - **Edit**→**T3 Mesh**→**Auto-adjust Nodes**: This tool uses a Laplacian smoothing algorithm to adjust the positions of interior nodes in the mesh to minimize distortion and equalize edge lengths. It can be used repeatedly to produce further adjustments.
-  - **Edit**→**T3 Mesh**→**Delete Unused Nodes**: This tool removes all nodes that are not part of an element.
-  - **Edit**→**T3 Mesh**→**Make All Elements CCW**: This tool renames nodes so that the nodes defining each element in the mesh increase in value when ordered counterclockwise.
-  - **Edit**→**T3 Mesh**→**Analyze Mesh**: This tool produces a statistical summary and analysis of the mesh in a pop-up window. The contents of the analysis can be saved into a text file by selecting **File**→**Save...** from the pop-up window's menu bar.
-  - **Tools**→**Compute Area**: This tool calculates the total area covered by the mesh. If the mesh is displayed in the Lat/Long coordinate system, the result is given in square metres.
-  - **Tools**→**Compute Volume**: This tool calculates the total displacement of the mesh, including the positive and negative volumes. If the mesh is displayed in the Lat/Long coordinate system, the result is in cubic metres.
-  - **Tools**→**Integrate Along Line**: This tool allows you to integrate the values along a line within the mesh. See "Extracting Integrals", on p. 103 for more details.

1.6.3.5 Resampling Data

Several types of data objects can be resampled, including Lines, LineSets, and Time Series. For information on resampling Time Series, see "Resampling Time Series" under TimeSeries Tools, on p. 107.

1.6.3.5.1 Resampling Lines and LineSets

To Resample a Line or LineSet:

1. Select an individual line from within a view or a lineset object from the workspace, and select the **Resample...** command from the shortcut menu. The following resample dialog will appear:

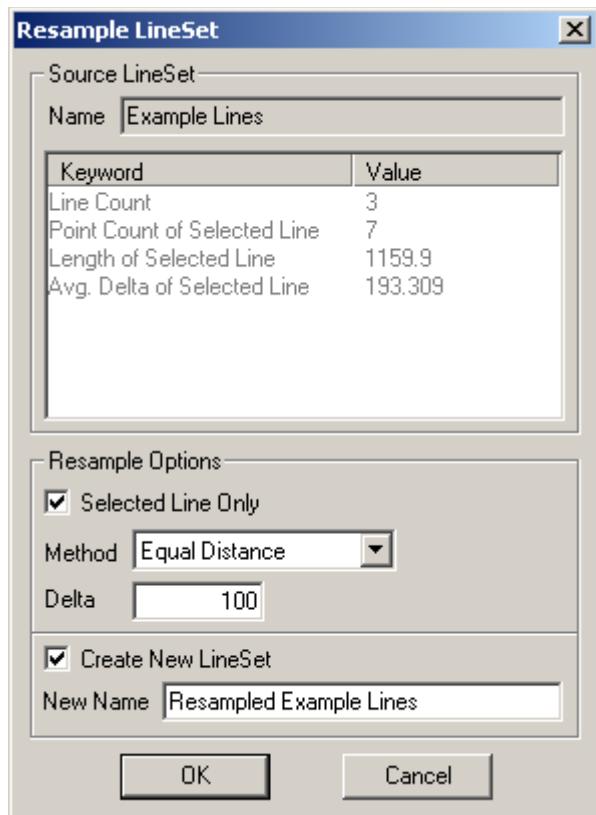


Figure 1.57: This dialog box is used to resample lines

The **Resample LineSet** dialog is divided into two parts: a **Source LineSet** section at the top; and a **Resample Options** section at the bottom.

The **Source LineSet** section contains data associated with the selected line or lineset object. Greyed text is read only.

2. Specify the resampling parameters within the **Resample Options** section.

If a line was selected from the view, you will have the option of resampling only that line or the entire lineset. There are three resampling methods to choose from: **Maximum Distance**, **Equal Distance**, and **Segment Count**.

- **Maximum Distance:** If this method is chosen, the 2D or 3D Line or Lines will be redrawn ensuring that the distance between each point does not exceed the **Delta** distance entered. Beginning at the first point on the 2D or 3D Line, if the distance from one point to the next is greater than the **Delta** value, a new point is inserted at the **Delta** distance. This procedure is repeated along the entire path of the 2D or 3D Line.
- **Equal Distance:** If this method is chosen, the 2D or 3D Line or Lines will be redrawn with the equal **Delta** distance between each points.

Note: The greater the **Delta** value, the greater the change in the shape will be from that of the original line.

- **Segment Count:** If this method is chosen, you will be prompted to enter a **Count** value instead of a **Delta**. The lines will be redrawn divided into **Count** segments; that is, the number of nodes, including the end points, will be equal to **Count**+1.

3. Once the resample options have been chosen, click . If the **Create New LineSet** box is checked, a new lineset will be created and added as a child under the source lineset in the workspace. The source lineset will remain unmodified. If the **Create New LineSet** box is unchecked, the source lineset will be overwritten with the changes.

1.6.3.6 Shifting Data Objects

Many data objects can be relocated, or shifted, changing their location. Objects that can be shifted include:

- Point Sets (*.pt2, *.xyz, *.pcl)
- Lines and Line Sets (*.i2s, *.i3s)
- Meshes (*.t3s, *.t3v, *.t3m, *.t3c, *.t4s, *.t4v)
- Grids (*.r2s, *.r2v, *.r2c)

To relocate a Data Object:

1. Select the Data Object within the WorkSpace or the View. In general, the entire Data Object will be relocated. To relocate individual points, see "Editing Points", on p. 75.

Note: If you've selected a Line within a View, only that Line will be relocated, not the entire Line Set. To relocate a Line Set, you must select it in the WorkSpace.

2. From the menu bar, select **Edit→Shift in X/Y....**

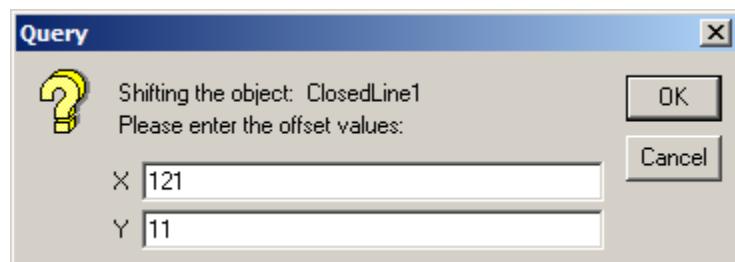


Figure 1.58: The Line Set *ClosedLine1* is being shift in both X and Y

3. Enter the offsets for the X and Y values and click .

1.6.4 Probing Data

Preliminary interpretation of the data can be done with the use of dynamic or static data probes. Static data probes display the data attributes associated with a specific component such as a particular line, cell or node of the object. Dynamic data probes, such as the live cursor, continually update the probe value as the cursor moves across the view.

1.6.4.1 Data Probes

Data probes display the data attributes associated with the individual components of an object. A component can be a node of a grid or mesh, a point from an xyz set, or a line segment from a shape file, line set object, or network. For objects with time-varying data, time series can be extracted from a particular component. See "Extracting Time Series" under Extracting Data, on p. 96, for more information.

To probe data:

1. Select a data item in a view so that it is highlighted in the Workspace. If no object is selected in the WorkSpace, EnSim scans the list of objects in the view, and selects and returns the data from the first object encountered under the mouse. If only one object is displayed in the view, it will be automatically selected.
2. Double click on a component (node, point, cell etc.) of an object in the view window. The object component selected for the data probe will be highlighted in magenta, and a popup window will appear. The popup window will display information about the node, such as the x and y coordinates and the node value. The node value is the current data attribute (see "Data Attributes" under Properties of Data Items, on p. 21).

Note: If you're examining a line or closed line, and the current coordinate system is LatLong, the Perimeter value will be given in metres and the Area value in square metres. These are calculated using the Great Circle Route Algorithm.

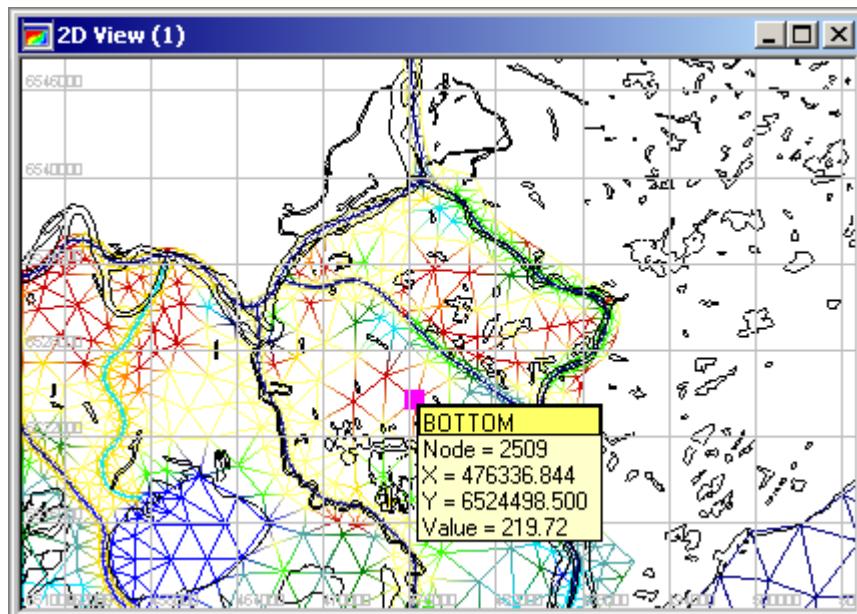


Figure 1.59: This data probe is displaying data about Node 2509

When a new object component is probed, the previous popup window disappears unless **Persistent Popups** is specified in the view window's **Properties**. All popup windows disappear when the view display is altered (i.e. when the object is "moved" in the view).

Properties of data probes may be edited in the **Display** tab of the view's **Properties** dialog box. There are three properties:

- **Persistent Popups:** When this has been selected, the popup windows of previous data probes remain open when others nodes are probed.
- **Extended Popup Info:** This option will display extended information, if any is available, in popup windows. For example, if an object with multiple data attributes is probed, all of the attributes will be displayed.
- **Show Probes:** This option highlights, using a green square outlined in magenta, locations from which time series have been extracted. Highlights will remain visible until the time series object is removed from the WorkSpace.

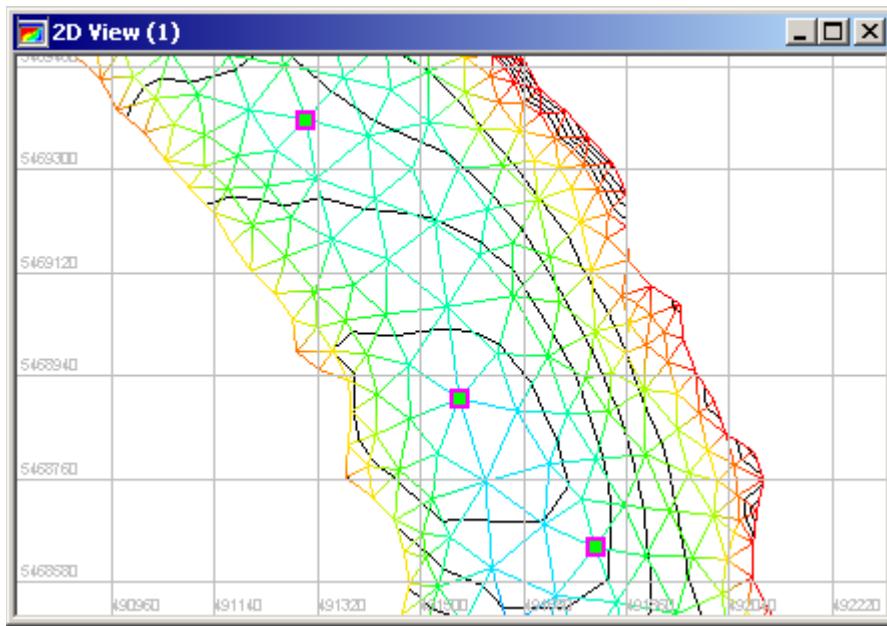


Figure 1.60: These nodes have had time series extracted from their data

Multiple probes can be viewed at the same time. To retain history of previous probes, choose **View→Selection info....** A window will appear in the view as shown below. If the **Extended Info** option is selected, the extended information will appear for each selected item

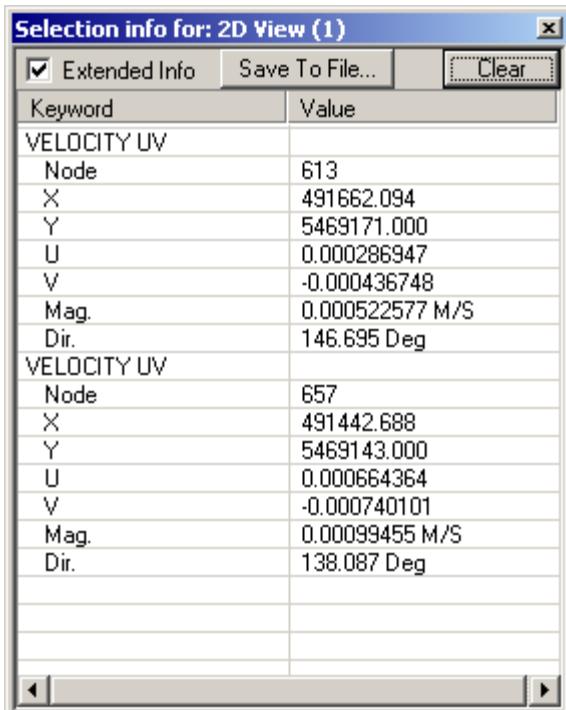


Figure 1.61: The Selection Info... window shows data from several probes

Any number of data points can have the attributes and their value viewed in this window. If the **Extended Popup Info** option is selected in the **Display** tab, the extended information will appear for each selected item.

Clicking on the **Save To File...** button opens a dialog box asking for the name of a file to which the data in the Selection Info window should be saved. The data will be saved as a simple text file.

1.6.4.2 The Live Cursor

The live cursor is a tool that displays attribute values on rectangular grids [<*.r2s, *.r2v, or *.r2c] and triangular meshes [<*.t3s, or *.t3v]. This value is interpolated at the position of the mouse cursor and is automatically updated as the cursor is moved over the select data object in a 2D view.

To use the live cursor, first ensure that the desired data object is selected, then click on the  button in the tool bar. Once the cursor is placed in the view, a box will appear next to the pointer, as shown in the image below. As the cursor is moved, the interpolated attribute value is updated and displayed next to the cursor. To turn off the live cursor, click the  button again or press the <Esc> key.

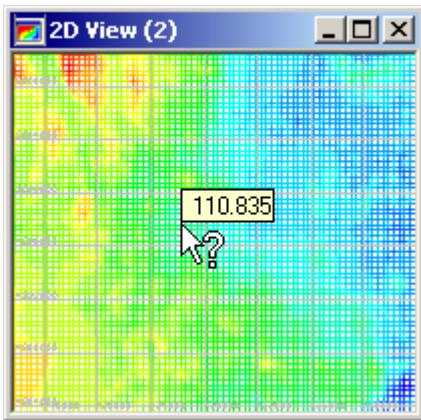


Figure 1.62: The Live Cursor shows interpolated data values

1.6.4.2.1 The Live Stream Lines Cursor

The live stream lines cursor is a tool that calculates and displays the path of a particle in a flow field. Stream lines are automatically enabled for rectangular vector grids [`*.r2v`] and triangular vector meshes [`*.t3v`]. The path is calculated at the position of the mouse cursor and is automatically updated as the cursor is moved over the select data object in a 2D view.

The options for the live stream lines cursor may be changed in the **Tools** tab of the 2D view's **Properties** dialog box.

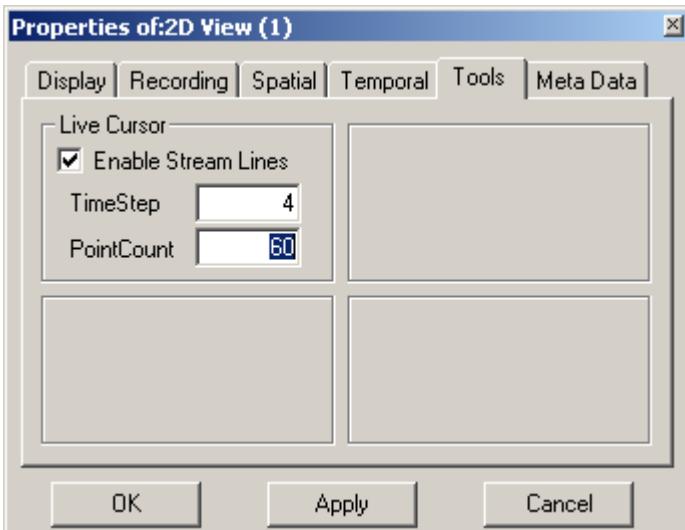


Figure 1.63: The Tools tab contains the live stream lines cursor options

- **Enable Stream Lines:** Checking this box enables stream lines. Unchecking the box enables the general live cursor [see above: "The Live Cursor" under Tools, on p. 85].
- **TimeStep:** The timestep used for calculating each point of the flow path.
- **PointCount:** The maximum number of points within the calculated flow path.

Note: For a vector grid or mesh with units of m/s, a TimeStep of 4 and a PointCount of 60 would compute a projected flow path over 240 seconds (4 times 60) using the velocities of the current frame.

To use the live stream lines cursor, ensure that the desired vector data object is selected and the Stream Lines are enabled, then click on the  button in the tool bar. Once the cursor is placed in the view, a projected flow path will be computed and displayed. As the cursor is moved, the projected flow path will be automatically updated. To turn off the live stream lines cursor, press the  button again or the <Esc> key.

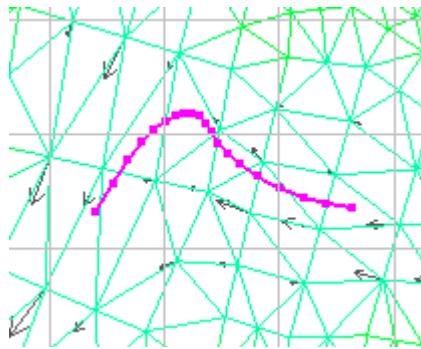


Figure 1.64: The Live Stream Lines Cursor shows a projected flow Path

Stream Lines can also be saved and added to 2D Line Sets [*.i2s].

To save stream lines:

1. While using the live stream lines cursor within the 2D view, right-click and select **Save StreamLine** from the shortcut menu. A dialog will appear which allows you to choose a name for the 2D Line Set.
2. Enter the name of the object to which the StreamLine will be added. If you use an existing object, the line will be added to the lineset.

The saved stream line will be added to the StreamLine set and the set is automatically added to the 2D view. Further stream lines may be added to this set (assuming the name is not changed) by repeating the above process.



Figure 1.65: Select a Line Set name for adding the stream line

1.6.4.3 The Ruler

The ruler is a simple tool that can be used to measure distances in a view. The units of measurement are the same as those of the view. For example, if the distance being measured

is on a 2D triangular mesh whose units are in metres, the displayed value of the measurement will also be in metres. The ruler tool can only be used in a 2D view.

To use the ruler, click on the  button in the tool bar. Click anywhere in the 2D view. This is the starting point, or zero, of the distance display. As the cursor is moved, the distance is updated and displayed next to the cursor. A mouse click will create a fixed point. A black dot will appear in the display and a yellow line connects this new point with the previous. This can be used to display distances along polylines (lines with multiple segments). The distance displayed will be the total distance from the starting point along the defined line. To turn off the ruler, click the  button, or press the <Esc> key.

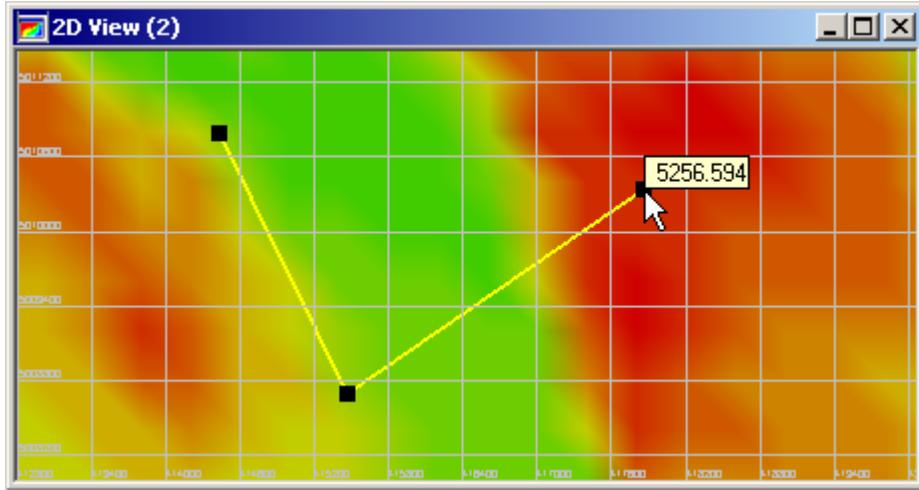


Figure 1.66: The ruler can be used to measure a polyline by creating fixed points

1.6.4.4 Computing Areas and Volumes

Two additional tools that are available to examine meshes and grids are the Area and Volume calculators. These tools can be used to determine the total area and displacement covered by grids and meshes.

To compute an area:

1. Select the data object in the WorkSpace or the View.
2. From the menu bar, select **Tools**→**Compute Area...**. The total area covered by the selected data object will be shown in a message window.

Note: EnSim uses a spherical polygon algorithm (Girards theorem) for computing areas in Lat/Long coordinate systems. Areas of Lat/Long polygons computed by EnSim will therefore differ from published areas which are typically computed in a projected coordinate system (such as Albers or Lambert Conformal)

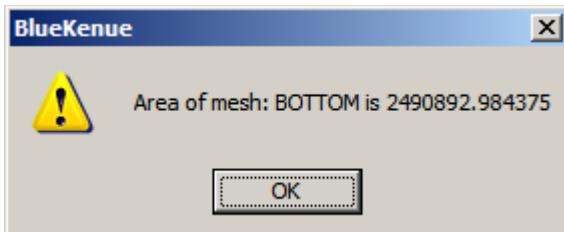


Figure 1.67: This mesh has an area of just over 2.49 km²

To compute a volume:

1. Select the data object in the WorkSpace or the View.
2. From the menu bar, select **Tools**→**Compute Volume....** The total displacement of the selected data object will be given in a message window.

The window includes both positive (i.e., all points with a value above zero) and negative (points below zero) displacements, as well as the net sum.

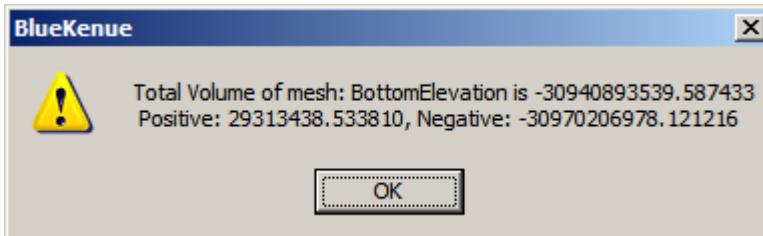


Figure 1.68: This bathymetric mesh has a net negative displacement

1.6.5 Extracting Data

Data extraction tools allow you to retrieve all data satisfying a particular criterion from an object in the WorkSpace. Take, for example, a triangular mesh object that has time-varying data. There is a new value at each node of the mesh for each time step. To retrieve the maximum value at each of these nodes, the Extract Surfaces tool can be used. By choosing the Temporal Maximums option, the maximum value in the time series at each node of the mesh is extracted. Data extraction tools also allow you to define isolines, to extract point values from a defined area within an object, or to calculate the residual vector from a mesh of time-varying vectors.

There are ten data extraction tools available in Kenuer, each of which is described below:

- Extract Surface
- Extract Residuals
- Extract IsoLines
- Extract Paths
- Extract Points
- Extract TimeSeries

- Extract Velocity Rose
- Extract an Attribute Table
- Extract Data from a Mesh
- Extract an Integral along a Line
- Extract XY Data from a Table

1.6.5.1 Extracting Surfaces

The *extract surface* tool can be used only with data that is in regular grid or triangular mesh format. Some of the sub-functions also require the data to vary with time. This tool selects a specific value for each node of a grid. The result is a 2D grid or mesh (a surface) identical to the parent grid or parent mesh, but with a particular, specified value at each node. In EnSim core, there are several options for the type of value that is to be extracted from each node of the parent grid or parent mesh:

1.6.5.1.1 Extracting Temporal Statistics

- **Temporal Maximums:** This option requires time-varying data. It finds the maximum value that occurred over the time series for each node in the grid or mesh.
- **Temporal Minimums:** This option requires time-varying data. It finds the minimum value that occurred over the time series for each node in the grid or mesh.
- **Temporal Ranges:** This option requires time-varying data. It calculates the range of values that occurred over the time series for each node in the grid or mesh.
- **Temporal Sums:** This option requires time-varying data. It calculates the sum of the values over the time series for each node in the grid or mesh.
- **Temporal Mean:** This option requires time-varying data. It calculates the mean of the values that occurred over the time series for each node in the grid or mesh.
- **Temporal StdDev:** This option requires time-varying data. It calculates the standard deviation of the values that occurred over the time series for each node in the grid or mesh.

To extract temporal statistics as a surface:

1. Select the data item from which the surface is to be extracted.
2. Select **Tools**→**Extract Surface**. Then choose the option for the type of value to be extracted from each node: **Temporal Maximums...**, **Temporal Minimums...**, **Temporal Ranges...**, **Temporal Sums...**, **Temporal Mean...**, or **Temporal StdDev...**
3. A Range Query dialog will appear. Enter the range of frames to be scanned for the specified values. The default is the total number of frames in the time series.



Figure 1.69: This dialog asks for the range of frames to be scanned

The new surface will be created as a child of the original object, and can be saved in the same format as the original grid or mesh file.

1.6.5.1.2 Extracting Slopes

The slope of a regular grid or triangular mesh can be extracted as a surface. The slope value is calculated at each node based on the elevation of that node and that of its neighbours. The units for slope may be either degrees or percent.

To extract the slopes, select the grid or mesh in the WorkSpace and then select **Tools**→**Extract Surface**→**Slopes** from the menu bar.

There are three methods available for computing slope on regular grid:

- **Finite Difference (8 neighbour):** The slope is computed at each node using the elevations of the node's eight neighbours. A 3rd-order finite difference method is used.
- **Finite Difference (4 neighbour):** The slope is computed at each node using the elevations of the node's neighbours in the four cardinal directions only. A 2nd-order finite difference method is used.
- **Steepest Descent:** The slope at each node is the steepest downhill slope found among the eight neighbours. If a node has a lower elevation than its neighbours, a slope value of zero is set for that node.

There is one method available for computing slope on a triangular mesh:

- **T3 Element Average:** The slope of each node of the mesh is computed as the average slope of all elements connected to the node. The slope of each element is calculated using cross products.

Once the slope option has been chosen, a Query dialog will appear. Select the units for the extracted slope: Degrees or Percent.:.



Figure 1.70: This dialog asks for the units of the extracted slope

The new slope surface will be created as a child of the original object, and can be saved in the same format as the original grid.

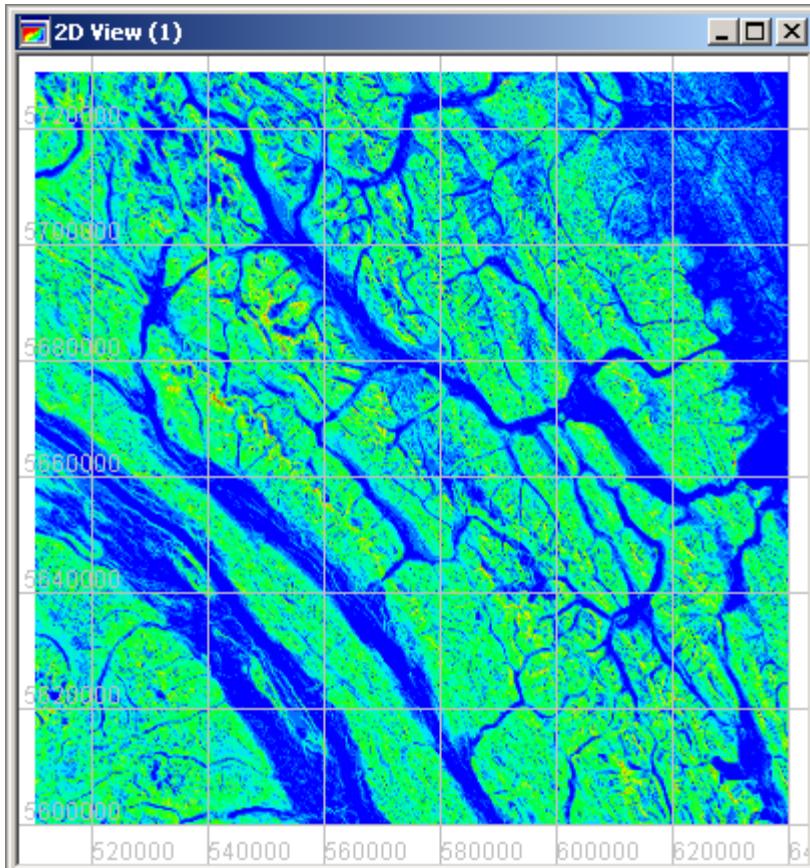


Figure 1.71: Slopes extracted using the eight neighbour finite difference option, shown in a 2Dview

1.6.5.1.3 Extracting Aspects

The aspect of a regular grid or triangular mesh can be extracted as a surface. The slope value is calculated at each node based on the elevation of the node and that of its neighbours. The units for aspect is degrees. North facing aspect is zero degrees with aspect increasing from 0 to 360 degrees in the clockwise direction (i.e. east facing is 90 degrees, south facing is 180 degrees, and west facing is 270 degrees).

To extract the aspects, select the grid or mesh in the WorkSpace and then select **Tools**→**Extract Surface**→**Aspects** from the menu bar.

There are two methods available for computing aspect on regular grid. They are as follows:

- **Finite Difference (8 neighbour):** The aspect is computed at each node using the elevations of the node's eight neighbours. A 3rd-order finite difference method is used.
- **Finite Difference (4 neighbour):** The aspect is computed at each node using the elevations of the node's neighbours in the four cardinal directions only. A 2nd-order finite difference method is used.

There is one method available for computing aspect on a triangular mesh:.

- **T3 Element Average:** The aspect of each node of the mesh is computed as the average aspect of all elements connected to the node. The aspect of each element is calculated using cross products.

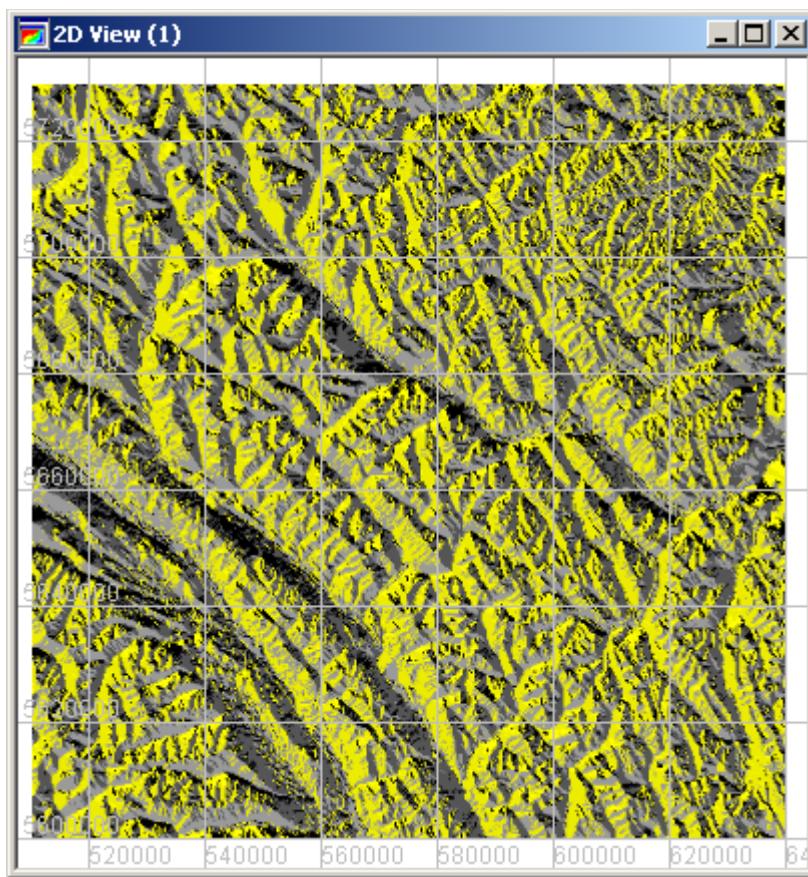


Figure 1.72: Aspects extracted using the eight neighbour finite difference option, shown in a 2Dview

1.6.5.1.4 Extracting Curvatures

The curvature of a regular grid can be extracted as a surface. Curvature values are calculated based on the slope and aspect and it's units are expressed as radians per metre. At the node, a positive value indicates a convex shape; a negative value indicates a concave shape; and a zero value indicates a flat shape.

To extract curvature, select the grid in the WorkSpace and then select **Tools**→**Extract Surface**→**Curvatures** from the menu bar.

There are four methods available for computing curvature on regular grid:

- **Profile Curvature:** a measure of the rate of change of slope in the direction of the slope. Profile curvature describes the shape of the surface in the direction of the slope.
- **Plan Curvature:** a measure of the rate of change of aspect along an elevation contour. Plan curvature describes the shape of the surface perpendicular to the direction of the slope.
- **Tangential Curvature:** the product of the Plan Curvature and the sine of the slope. Tangential curvature describes the shape of the surface in a vertical plane perpendicular to the direction of the slope.
- **Total Curvature:** a measure of the curvature of the surface in all directions. Note: a zero value may mean that a concave shape in one direction offsets a convex shape in another direction at that particular node.

Note: There are no methods available for computing curvatures on a triangular meshes.

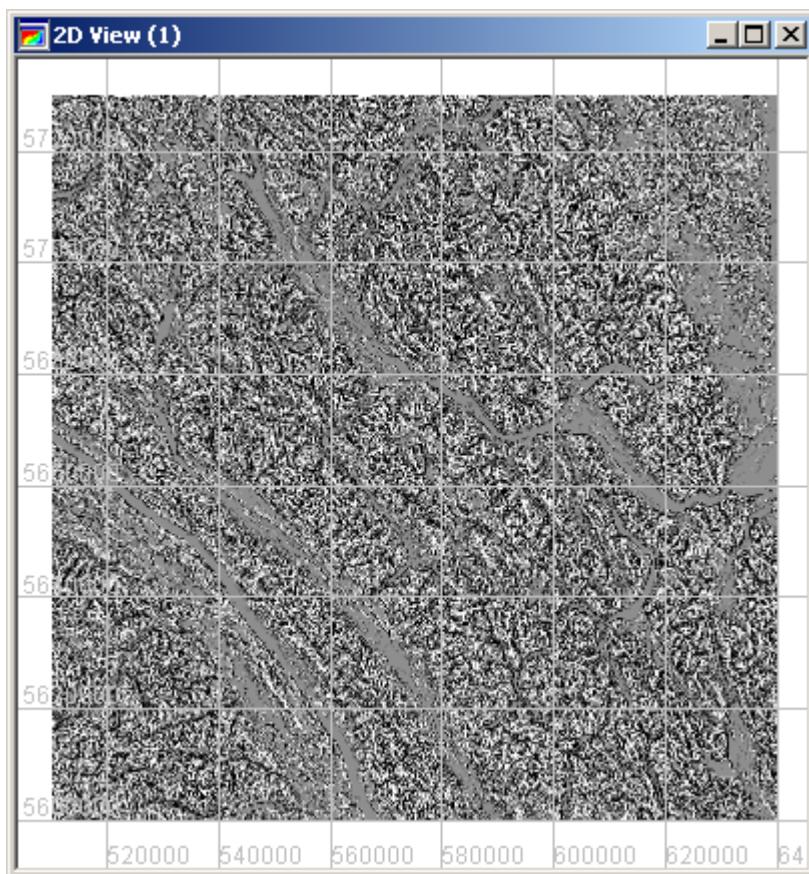


Figure 1.73: Profile curvature shown in a 2Dview

1.6.5.2 Extracting Residuals

The *extract residuals* tool calculates the resultant, or residual, vector from all vectors occurring at a grid or mesh point over time for each node of a grid or mesh.

To extract residuals:

1. Select the object for which the residuals are to be calculated. The object must be a rectangular grid or triangular mesh and the data must be vector data. Appropriate objects in the WorkSpace will have one of these two icons:  or .
2. Select **Tools**→**Extract Residuals**.

The residual vectors will be created as a child of the original grid or mesh, and can be saved in the same format as the parent file.

1.6.5.3 Extracting Isolines

The *extract isolines* tool creates one or more lines connecting nodes of specific values.

To extract an isoline:

1. Select the object in the WorkSpace for which the isolines are to be extracted.
2. Select **Tools**→**Extract Isolines**, and then select **Single Isoline...** or **Multiple Isolines...**
 - For the **Multiple Isolines...** option, the isolines will be created using the levels specified under the data item's **Colour Scale** tab in its **Properties** dialog.
 - For the **Single Isolines...** option, a Query Dialog will appear. Enter the value for which the isoline is to be extracted. The range of possible values will be provided in the dialog.

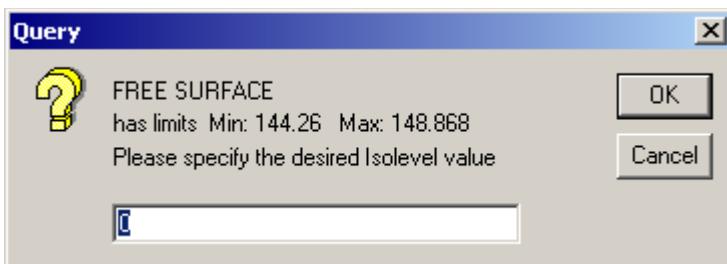


Figure 1.74: The limits of the range of values for a single isoline are provided

A single isoline can also be extracted using the shortcut menu of the data item. The isolines are added to the WorkSpace as children of the original object, and can be saved as an *.i2s, *.i3s, *.xyz, *.shp, or *.mif file.

1.6.5.4 Extracting Paths

The *extract path* tool can be used only with time-varying parcel data. A set of lines is created following the path of a selected parcel from the beginning to the end of the dataset.

To extract a path:

1. Select the object in the WorkSpace for which the path is to be extracted.
2. Select the parcel in the object for which the path is to be extracted.
3. Select **Tools→Extract Path....** A new path file will appear added to the WorkSpace as a child of the object.

1.6.5.5 Extracting Points

A point set can be extracted from any object that is composed of point data with z, or attribute, data, including grids, meshes, point data (xyz), and cross-sections (3D lines).

To extract points from a data item:

1. Select the data item in the WorkSpace from which the points are to be extracted.
2. Draw a closed polyline around the region of the data item from which the points are to be extracted. See "Drawing Lines and Closed Polylines" under Creating New Data Items, on p. 69, for more details.
3. Select **Tools→Extract Points....** Choose the appropriate (newly drawn) polygon from the list. A new point set will be added to the WorkSpace as a child of the data item.

1.6.5.6 Extracting Time Series

Time series can be extracted from any object in the WorkSpace having data that varies with time. The resulting time series can be viewed in a 1D window.

To extract a time series:

1. Select the data item in the WorkSpace from which the time series is to be extracted. If you will be extracting a time series at multiple points, you must create a point set before continuing. See "Drawing Points" under Creating New Data Items, on p. 68, for more details.
2. Select **Tools→Extract TimeSeries.**
3. Select **At Selected Point, At Multiple Points..., Constrained By..., or Along a Line....**
 - If **At Selected Point** is chosen, a time series is extracted at the currently selected component's location. Probe the data at one node. See "Probing Data" under Tools, on p. 82, for more details.
 - If **At Multiple Points...** is chosen, a point set must have been created. See "Drawing Points" under Creating New Data Items, on p. 68, for more details.

Once the new point set is created, select **At Multiple Points....** A list of available point sets from which the time series may be extracted within the current object are shown in the dialog window that appears. Once a point set is selected, spatial interpolation is used to extract a time series for each point. The series are then displayed in the WorkSpace.

- If **Along a Line...** is chosen, a line must have been created. See "Drawing Lines and Closed Polylines" under Creating New Data Items, on p. 69, for more details.

Once the new line is created, select **Along a Line...**. A list of available line sets from which the time series may be extracted within the current object are shown in the dialog window that appears. Once a line set is selected, spatial interpolation is used to generate time series for each point along the line. These series are used to create a surface which is a special time series known as a time grid (*.ts5). See "Time Series [ts1 / ts2 / ts3 / ts4 / ts5]", on p. 288 for more information on this file type.

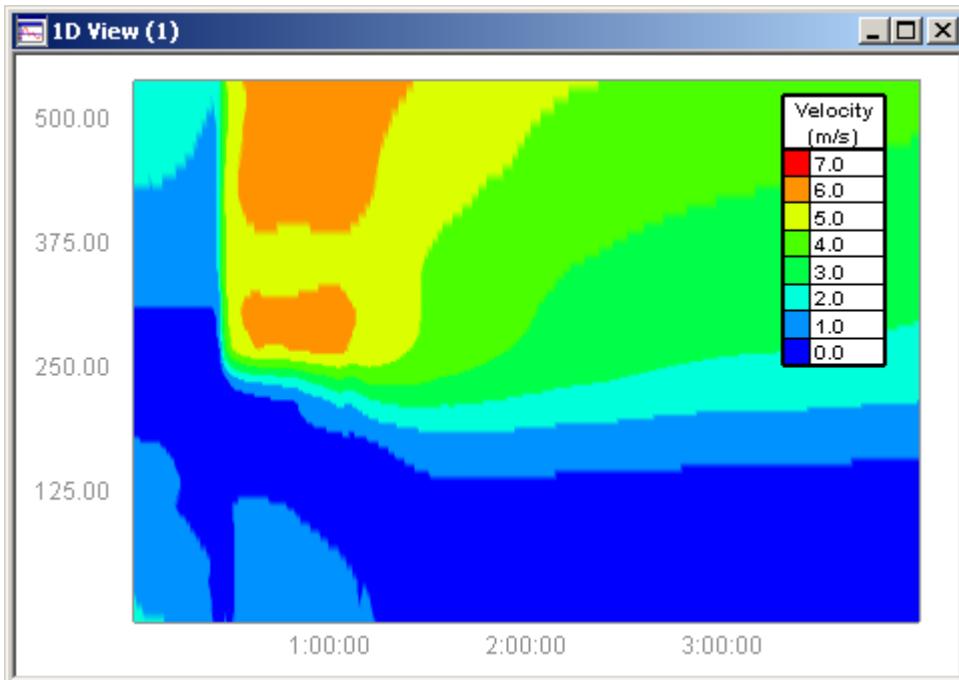


Figure 1.75: This time grid visualized in a 1D view

The above figure illustrates the time grid. The y-axis measures the distance along the line and the x-axis denotes the time. The colours as defined in the legend represent the velocities along the line as they span through time.

- If **Constrained By...** is chosen, the following dialog will appear:

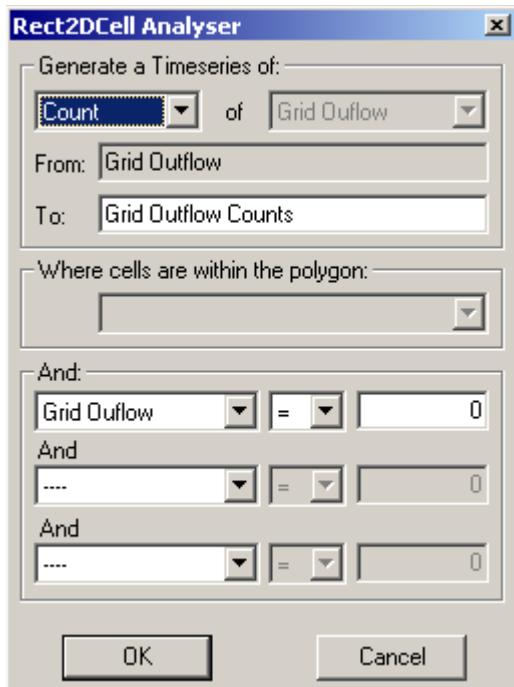


Figure 1.76: This dialog gathers criteria for the constraint

A constrained time series allows the use of Boolean operators and Aggregation for extracting time series from parcel files.

- **Generate a Timeseries of:** This option must be chosen from **Count**, **Sum**, **Min**, **Max**, and **Average**. The **Count** option counts the number of points defined by the Boolean operators in the **And:** box, below. The **Sum**, **Min**, **Max**, and **Average** options complete the specific operation on a specific attribute.
- **From:** This option shows the source object which contains the data to be examined.
- **To:** This option provides a name for the time series that is to be created.
- **Where cells are within the polygon:** This option lists any closed polylines within the data item, which may be used to provide a spatial constraint. The word **cells** in the title of this box may be different, depending on the type of data item.
- **And:** Each of these three options lists the attributes within the data item and the available Boolean operators (=, <, and >). The third column allows you to enter a value to define the criterion.

The time series data item will be created and displayed in the WorkSpace as a child of the originating data item. The time series may be saved in *.ts# format, where # ranges from 1 to 5 depending on the type of time series data. See "Time Series [ts1 / ts2 / ts3 / ts4 / ts5]", on p. 288, for details on types of time series data.

1.6.5.7 Extracting a Velocity Rose

A velocity rose can be extracted from any vector time series (.ts2 or .ts4) in the WorkSpace. The velocity rose is a statistical representation of the frequency of occurrence of speed and direction. The velocity rose plot can be viewed in a Polar view (see "The Polar View Window", on p. 37) or a 1D view window.

To extract a velocity rose:

1. Select the vector time series object from the WorkSpace for which the velocity rose is to be extracted.
2. Select **Tools**→**TimeSeries**→**Compute Velocity Rose**

The following dialog will appear:

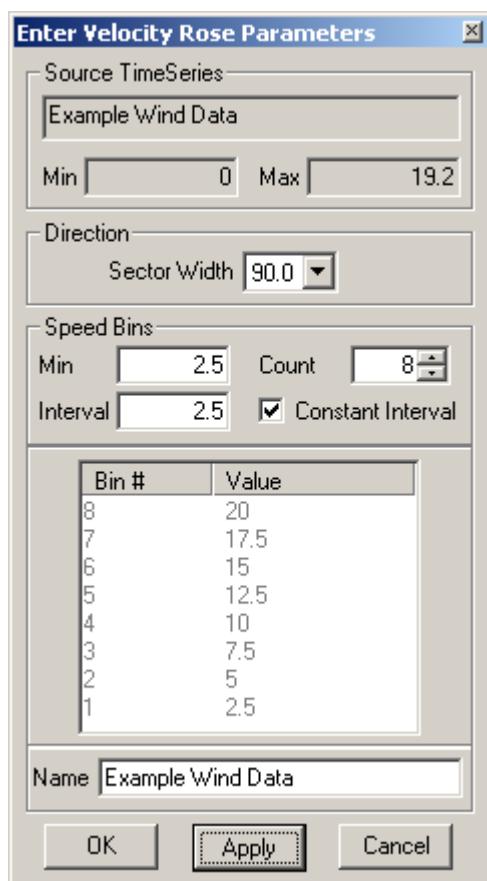


Figure 1.77: The Velocity Rose Parameters dialog

The velocity rose dialog parameters are described below:

- **Source TimeSeries:** The name of the source time series as well as the **Min** and **Max** values are shown.
- **Direction:** Select a **Sector Width** (22.5, 30, 45, or 90 degrees). This value defines the number of directional sectors. For example, a **Sector Width** of 90 degrees will define four 90 degree sectors for computing the velocity rose.

Note: The first sector is always centred on 0 degrees. For example, with a sector width of 90 degrees, the first sector will span from NorthWest to NorthEast (-45 degrees to 45 degrees).

- **Speed Bins:** Set up the bins using the following parameters:

- **Min:** Specify the upper limit used by the first bin.
- **Count:** Specify the number of bins.
- **Interval:** Specify the width of the bins.

Click  to update the list containing bin # and values based on the selected the Min, Count, and Interval.

- **Constant Interval:** If checked, the bin widths will remain constant using the **Interval** value. If unchecked, the minimum value for each bin becomes editable in the list within the dialog.
- **Name:** Enter the name to be used by the extracted velocity rose object.

Click  to create and display the velocity rose object in the WorkSpace as a child of the originating vector time series. The velocity rose can be saved as an ASCII .vr1 file. See "Velocity Roses [vr1]", on p. 295 for more information on this file type.

1.6.5.8 Extracting an Attribute Table

Attribute tables can be used to show the data from a time series, a table, or any multiattribute object in tabular format. These files can then be saved as tables or comma-separated text files.

To Extract an Attribute Table

1. In the Workspace, select the data object whose attribute table you'd like to examine.
2. Right-click on the object and select **Show Attribute Table...**, or select **Edit→Show Attribute Table...** from the menu bar.

The attribute table will be created as a child object of the data object and will be displayed. The attribute table can be saved by selecting it and clicking the  button in the toolbar, or selecting **File→Save** from the menu bar. The attribute table can be saved as either a .tb0 or .csv file. See "Tables [tb0]", on p. 293 for more information on Table files.

Note: After you've extracted an Attribute Table, the contents of the table will not be updated if the source data object changes. To refresh the data in the table, you must repeat the extraction process.

Cities_7.5m Attributes					
id	X	Y	NAME	NTS50	POP91
441	-119.433	49.9302	Kelowna	082E14	75953
442	-122.793	53.9311	Prince George	093G15	69653
443	-89.3176	48.4529	Thunder Bay	052A06	113946
444	-106.651	52.1606	Saskatoon	073B02	186067
445	-83.022	42.3053	Windsor	040J06	191435
446	-80.5136	43.4369	Kitchener	040P08	168282
447	-79.2869	43.2007	St. Catharines	030M03	129300
448	-78.8896	43.9204	Oshawa	030M15	129344
449	-73.6629	45.57	Montréal	031H12	1017...
450	-79.9197	43.2681	Hamilton	030M04	318499
451	-81.2912	43.0086	London	040I14	311620
452	-79.5887	43.6555	Brampton	030M12	234445
453	-114.062	51.0738	Calgary	082001	710677
454	-123.15	49.2676	Vancouver	092G03	471844
455	-123.418	48.46	Victoria	092B06	71228
456	-104.65	50.478	Regina	072I07	179183
457	-79.4193	43.6738	Toronto	030M11	635395
458	-75.7271	45.4013	Ottawa	031G05	313987
459	-71.3375	46.8882	Québec	021L14	167517
460	-66.7034	45.9526	Fredericton	021G15	46466
461	-63.1679	46.2913	Charlottetown	011L03	15396
462	-63.6929	44.6157	Halifax	011D12	114455
463	-52.8081	47.6034	St. John's	001N10	104659
464	-97.2476	49.9067	Winnipeg	062H14	615215
465	-113.575	53.5473	Edmonton	083H11	616741
466	125.725	50.7400	Whitehorse	105D11	17025

Figure 1.78: This Attribute Table is from the 1:7,500,000 Cities Base Map

1.6.5.9 Extracting Data From a Mesh

There are several tools that allow you to extract portions of a mesh.

1.6.5.9.1 Extracting a Subset of a Mesh

To extract a subset of a mesh:

1. Create or open a closed line set that encompasses the area that you wish to isolate. The closed line must completely contain all nodes belonging to the elements to be extracted. See "Drawing Lines and Closed Polylines", on p. 69 for more information on creating a closed line.
2. In the WorkSpace, select the mesh that contains the elements to be extracted.
3. From the menu bar, select **Tools**→**T3 Mesh**→**Extract Subset...**
4. In the **Extract Mesh Subset From...** dialog, select the closed line set from the **Where Elements are within the Polygon** drop-down menu.
5. Select the frames from which the subset will be extracted by selecting a choice from the **Frame Option** drop-down menu. If the mesh doesn't contain multiple frames, this option will be unavailable.

- **Current Frame:** Selects only the frame which is currently visible, as indicated in the Current Frame box. To change the current frame, use the animation toolbar. See "Animation", on p. 61 for more information.
- **Range of Frames:** This option lets you choose a temporal subset of frames to extract from. Enter the first and last frames in the **Start Frame** and **End Frame** boxes, respectively. If the Start Frame comes after the End Frame, only the Start Frame will be extracted.
- **All Frames:** This option extracts a subset from all frames of the parent mesh.

6. Click **OK** and enter a filename for the subset. The subset will be displayed in the WorkSpace as a child object of the parent mesh.

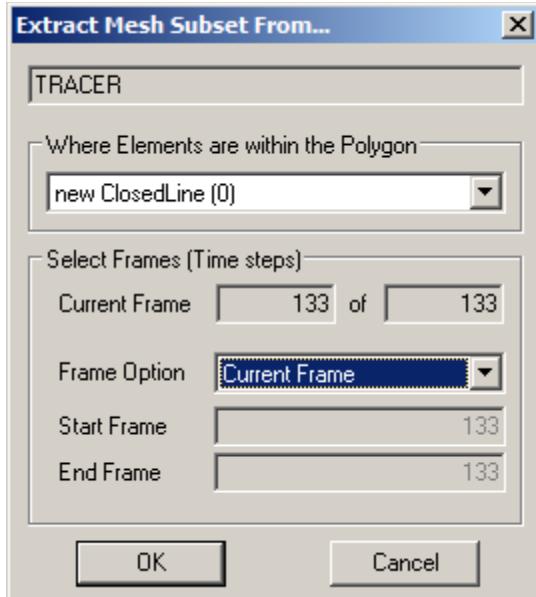


Figure 1.79: This dialog is used to extract a subset of a mesh

1.6.5.9.2 Extracting the Edge of a Mesh

To extract the edge of a mesh:

1. In the WorkSpace, select the mesh whose edge you wish to extract.
2. From the menu bar, select **Tools**→**T3 Mesh**→**Extract Edges (Shorelines)**. The nodes that form the edge of the mesh will be extracted as a 2D line set, which will be created as a child object of the parent mesh.

1.6.5.9.3 Extracting Edge Lengths From a Mesh

To extract the edge lengths of a mesh:

1. In the WorkSpace, select the mesh whose edge lengths you wish to extract.
2. From the menu bar, select **Tools**→**T3 Mesh**→**Extract EdgeLengths**. The resultant mesh will be created as a child object of the parent mesh. The EdgeLengths mesh contains the same nodes, but the value of each node is equal to the average of its distance from its neighbours.

1.6.5.10 Extracting Integrals

The **Integrate along Line...** tool allows you to extract the rates of change of a variable along a line within a view.

To extract an integral along a line

1. Using the **New Open Line** tool, create a line within the View that indicates the line along which you'd like to integrate. See "Drawing Lines and Closed Polylines", on p. 69 for more information.
2. Select the line in the WorkSpace.
3. From the menu bar, select **Tools**→**Integrate alone Line...** or click the  button on the T3 Mesh Toolbar. This will open the Integration dialog box.

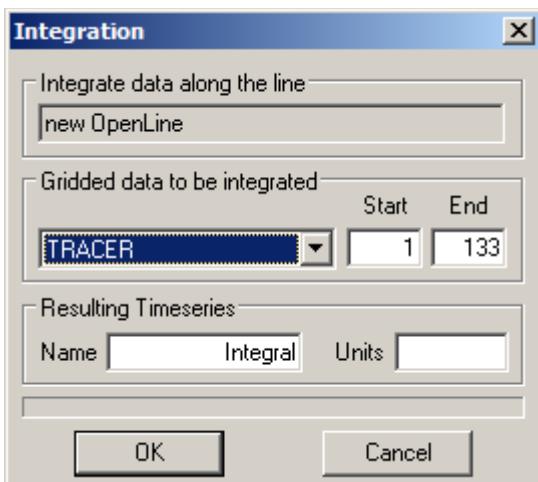


Figure 1.80: This dialog box is used when integrating

4. In the drop-down menu, select the data object containing the data you'd like to integrate. This box will list all time-variant objects that are currently loaded.
5. Enter the numbers of the **Start** and **End** frames.
6. Enter a **Name** and appropriate **Units** for the new data object. These will be stored in the new ts3 Time Series.
7. Click . The new Time Series will be created and listed in the WorkSpace as a child object of the source data object.

1.6.5.11 Extracting XY Data From a Table

Any table can be used as the source of an XY Data object comparing any two columns from the table.

To extract XY data from a table:

1. Select the table within the WorkSpace. From the shortcut menu, select **Extract XY Data....**

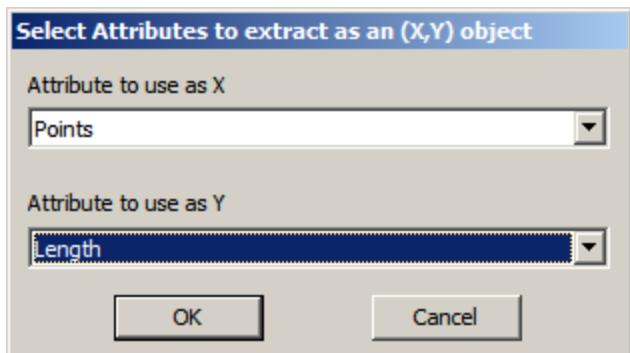


Figure 1.81: This XY Data object will compare Points to Length

2. From the drop-down menus in the **Select Attributes to extract as an (X,Y) object** dialog box, select the two columns that will be compared. Click **OK**. The XY Data object will be created as a child of the Table object.

1.6.6 TimeSeries Tools

Several tools are available in Kenue that allow you to create, analyze and compare various types of Time Series. For information on creating a Time Series from a time-variant data object, see "Extracting Time Series", on p. 96.

1.6.6.1 Editing Time Series

To launch the time series editor dialog, either select a point in the time series or the time series object itself, and select **Edit...** from the shortcut menu. If a point is selected, that point will be highlighted in the **Data Points** list box. An example of the time series editor dialog with the selected point highlighted is shown below.

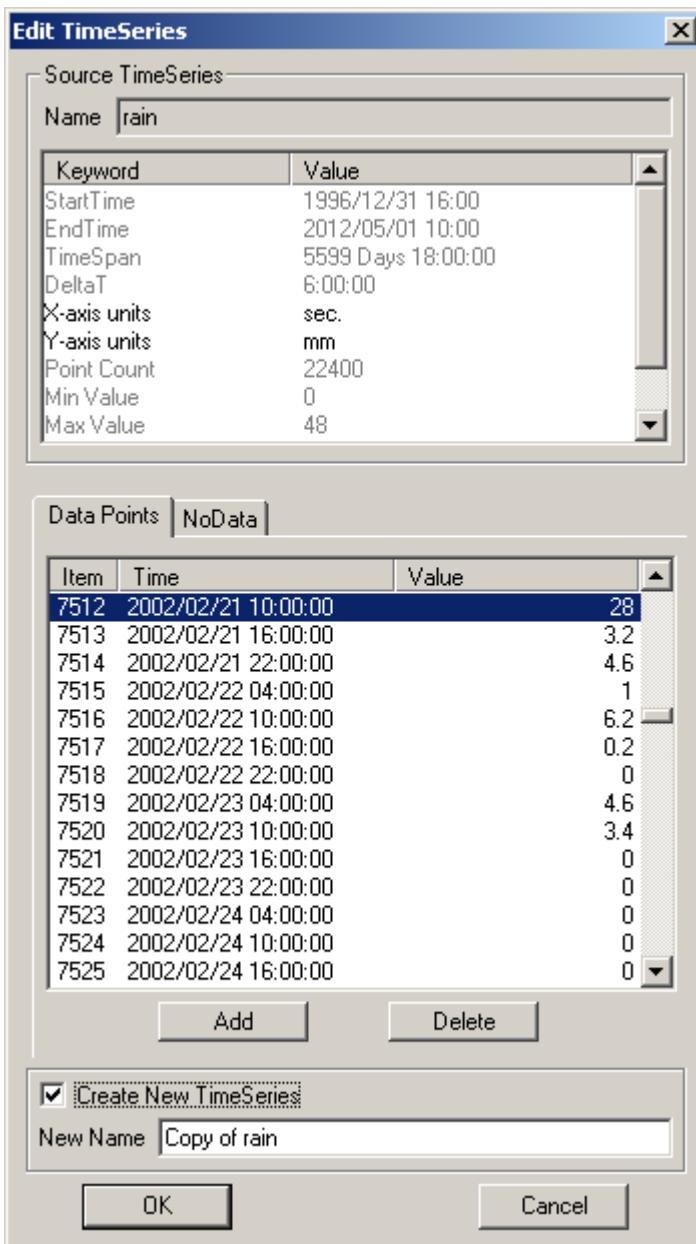


Figure 1.82: This dialog box is used to edit time series

The time series editor dialog is divided into three parts: the **Source TimeSeries** data section at the top; the two edit pages (**Data Points** and **NoData**) in the middle; and the option for creating a new timeseries or overwriting the source time series at the bottom.

The **Source TimeSeries** section contains data associated with the time series object. Some fields such as **X-axis units** and **Y-axis units** are editable. Greyed text is read-only.

The **Data Points** page lists the item number, the time, and the value for every point in the time series object. Both the time and the value are editable fields. Data points may be added by using the **Add** button. Selected data points may be removed by using the **Delete** button.

Note: Although added points appear at the bottom of the point list, the points are automatically sorted by time when you click **OK**.

The **NoData** page gives you several options for replacing **NoData** values. **NoData** values are points in the time series that have an associated time but no valid value. In this case, a user specified **NoData** value is set for that point. An example of a **NoData** page is illustrated below.

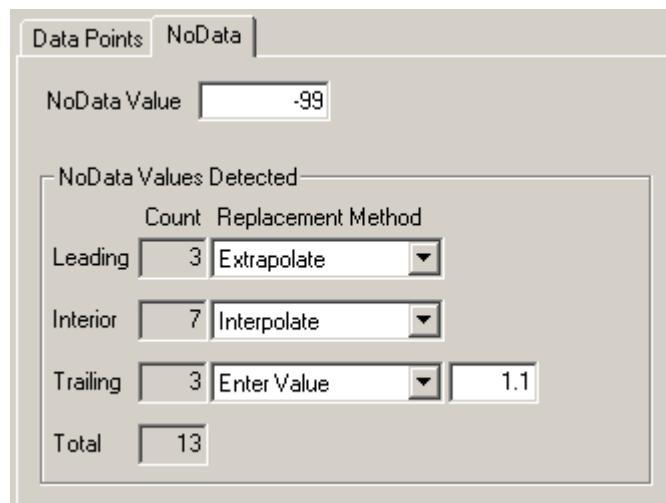


Figure 1.83: The time series NoData page

The **NoData** page allows for setting or the editing of the **NoData** value as well as the detection and tools for **NoData** value replacement. The replacement tools are as follows:

Leading NoData values are **NoData** values found at the beginning of the time series.

Replacement methods for leading **NoData** values include:

- **None Selected:** Do nothing.
- **Extrapolate:** Extrapolate linearly from the first valid point values found.
- **First Valid Value:** Replace with the first valid point value found.
- **Enter Value:** Enter the replacement value. This option is available for ts1 & ts3 objects only.
- **Enter Mag&Dir Value:** Enter the replacement magnitude and direction values. This option is available for ts2 & ts4 objects only.

Interior NoData values are **NoData** values found anywhere in a time series as long as they follow and precede a valid values. Replacement methods for interior **NoData** values include:

- **None Selected:** Do nothing.
- **Interpolate:** Interpolate linearly using the last and next valid point values found.
- **Last Valid Value:** Replace with the last valid point value found.
- **Next Valid Value:** Replace with the next valid point value found.
- **Enter Value (ts1 & ts3 only):** Enter the replacement value.
- **Enter Mag&Dir Value (ts2 & ts4 only):** Enter the replacement magnitude and direction values.

Trailing NoData values are **NoData** values found at the end of the time series. Replacement methods for trailing **NoData** values include:

- **None Selected:** Do nothing.
- **Extrapolate:** Extrapolate linearly from the last valid point values found.
- **Last Valid Value:** Replace with the last valid point value found.
- **Enter Value:** Enter the replacement value. This option is available for ts1 & ts3 objects only.
- **Enter Mag&Dir Value:** Enter the replacement magnitude and direction values. This option is available for ts2 & ts4 objects only.

Once the edits have been completed, press **OK**. If the **Create New TimeSeries** box is checked, a new time series will be created and added as a child under the source time series in the workspace. The source time series will remain unmodified. If the **Create New TimeSeries** box is unchecked, the source time series will be overwritten with the changes.

1.6.6.2 Resampling Time Series

Select a time series from within a view or a time series object from the workspace, and select the **Resample...** command from the shortcut menu. The following resample dialog will appear:

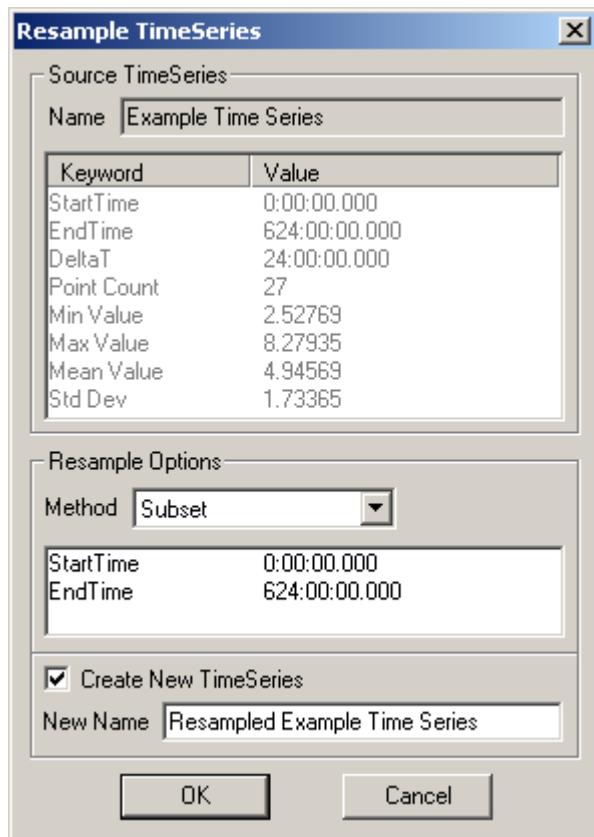


Figure 1.84: This dialog box is used to resample time series

The Resample TimeSeries dialog is divided into two parts: a **Source TimeSeries** section at the top; and a **Resample Options** section at the bottom.

The **Source TimeSeries** section contains data associated with the time series object. Greyed text is read only.

You can specify resampling parameters within the **Resample Options** section.

There are several resampling methods to choose from:.

- **Subset**: Any time series points found between and including the resample **StartTime** and **EndTime** values will be subsetted.
- **Temporal Shift**: All time series points will be shifted to match the new resample **StartTime** value. For example, if the source **StartTime** is at 12:00:00.000 (12 hours) and you enter a resample **StartTime** of 18:00:00.000 (18 hours), then all time series points will be shifted forward by 6:00:00.000 (6 hours).
- **Linear Interpolation**: A new time series is generated by linearly interpolating the source time series using the resample **DeltaT**. The timespan of this interpolated time series is defined by the resample **StartTime** and **EndTime** values (default to the full timespan).
- **Cubic Spline** (ts1 only): A new time series is generated by the cubic spline method using the resample **DeltaT**. The timespan of this splined time series is defined by the resample **StartTime** and **EndTime** values (default to the full timespan). Note: the cubic spline method requires a source time series with a constant **DeltaT**.
- **Interval Sums** (ts1 & ts3 only): A new time series is generated by summing all values found within each resample **DeltaT**. The timespan of this new time series is defined by the resample **StartTime** and **EndTime** values (default to the full timespan). Note: the **Interval Sums** method is not applicable for vector time series (i.e. directions).
- **Interval Means**: A new time series is generated by calculating the mean values found within each resample **DeltaT**. The timespan of this new time series is defined by the resample **StartTime** and **EndTime** values (default to the full timespan).
- **Daily Sums** (ts1 & ts3 with calendar dates only): A new time series is generated by summing all values found within each calendar day. The timespan of this new time series is defined by 0 hours of the day identified by the resample **StartTime** and 24 hours of the day identified by resample **EndTime**. Note: the **Daily Sums** method is not applicable for vector time series (i.e. directions) and requires calendar dates only.
- **Daily Means** (calendar dates only): A new time series is generated by calculating the mean values found within each calendar day. The timespan of this new time series is defined by 0 hours of the day identified by the resample **StartTime** and 24 hours of the day identified by the resample **EndTime**.
- **Monthly Sums** (ts1 & ts3 with calendar dates only): A new time series is generated by summing all values found within each calendar month. The timespan of this new time series is defined by 0 hours of the first day of the month identified by the resample **StartTime** and 24 hours of the last day of the month identified by the resample **EndTime**. Note: the **Monthly**

Sums method is not applicable for vector time series (i.e. directions) and requires calendar dates only.

- **Monthly Means** (calendar dates only): A new time series is generated by calculating the mean values found within each calendar day. The timespan of this new time series is defined by 0 hours of the first day of the month identified by the resample **StartTime** and 24 hours of the last day of the month identified by the resample **EndTime**.

Once the resample options have been chosen, click **OK**. If the **Create New TimeSeries** box is checked, a new time series will be created and added as a child under the source time series in the workspace. The source time series will remain unmodified. If the **Create New TimeSeries** box is unchecked, the source time series will be overwritten with the changes.

1.6.6.3 Computing Performance Statistics

The Compute Performance Statistics... tool can be used to compare two TimeSeries and produce statistics showing how closely the two series concur. This can be used to compare measured and observed results, for example.

To compute performance statistics:

1. In the WorkSpace, select one of the TimeSeries objects that you would like to compare.
2. From the menu bar, select **Tools**→**TimeSeries**→**Compute Performance Statistics...**

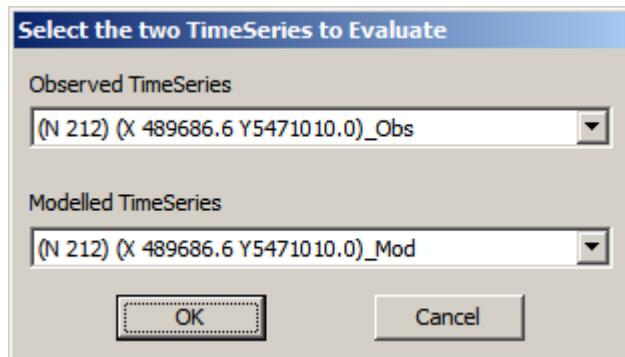


Figure 1.85: These Time Series will be compared statistically

3. From the drop-down menus, select the two TimeSeries objects that you'd like to compare.
4. Click **OK**. The results of the analysis will be shown in a text window.
 - Root Mean Squared
 - R-Squared
 - Nash-Sutcliffe Model Efficiency Coefficient
 - Relative Volume Error
 - Volumetric Efficiency
 - Mean Squared Log Error
 - Variance

- Bias
- Relative Bias
- Mean Squared Error
- Mean Absolute Error
- Mean Absolute Relative Error
- Mean Squared Relative Error
- Monthly Peak Differences (month-by month)
- Monthly Percent Error in Peaks (month-by month)
- Monthly Relative Volume Error (month-by month)

5. Save the contents of this window to a text file by selecting **File→Save...** from the text window's menu bar and providing a file name.

1.6.6.4 Computing a Flow Duration Curve

The Flow Duration Curve indicates how frequently a TimeSeries equals or exceeds a particular value, as a percentage of time.

To compute a flow duration curve:

1. In the WorkSpace, select the TimeSeries to be examined.
2. From the menu bar, select **Tools→TimeSeries→Compute Flow Duration Curve**, or select **Compute Flow Duration Curve** from the shortcut menu.

The Flow Duration Curve XY Data Object will be created and displayed in the WorkSpace as a child object of the parent TimeSeries, with the  icon and the suffix **_FDC**. The Flow Duration Curve object can be viewed in a 1D view and saved as a .xy file.

1.6.6.5 Computing a Cumulative Sum

To compute a cumulative sum:

1. In the WorkSpace, select the TimeSeries whose sum you'd like to compute.
2. From the menu bar, select **Tools→TimeSeries→Compute Cumulative Sum....**

The Cumulative Sum TimeSeries will be created and displayed in the WorkSpace as a child object of the parent TimeSeries, with the  icon and the suffix **_Sum**. The Cumulative Sum object can be viewed in a 1D View and saved as a .t3s file.

1.6.6.6 Computing an Integral

The integral value of a TimeSeries represents the sum of the area under the curve.

To compute the integral of a TimeSeries:

1. In the WorkSpace, select the TimeSeries to be examined.
2. From the menu bar, select **Tools**→**TimeSeries**→**Compute Integral...**. The integral will be displayed in a message window.

The message window indicates the name of the source TimeSeries, the number of data points used in the calculation, the start and end times of the TimeSeries, and the value of the integral.

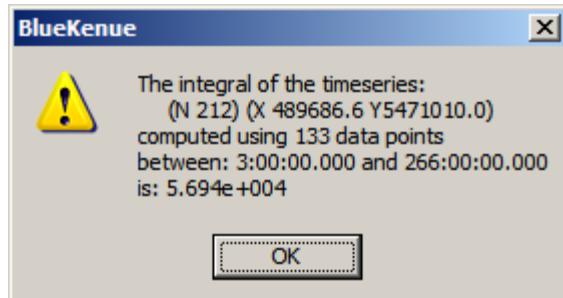


Figure 1.86: The integral of the TimeSeries is 56,940

1.6.6.7 Computing a Distribution

A probability density distribution curve can be computed from a time series. This curve displays the distribution of data for a record set. Only the records included in a temporal subset will be used for computing the distribution.

To create a distribution:

1. In the WorkSpace, select the TimeSeries to be examined.
2. Select **Tools**→**TimeSeries**→**Compute Distribution** from the menu bar, or **Compute Distribution** from the shortcut menu. The distribution will appear as a child of the selected time series, with the **_Distribution** suffix. A distribution object is represented by the  icon, and can be saved as a .dat file.

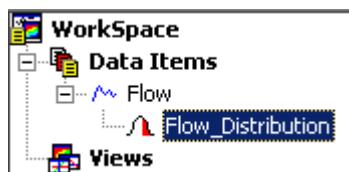


Figure 1.87: The distribution appears as a child of the time series

The **Data** tab from the **Properties** dialog of the distribution displays some useful information about the time series.

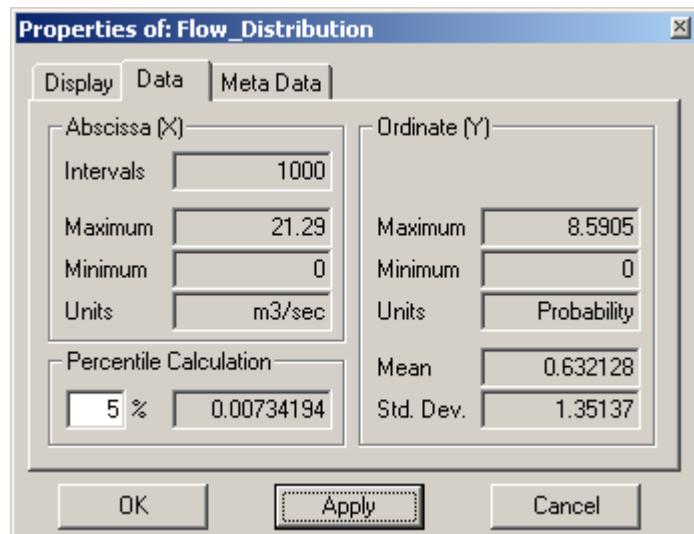


Figure 1.88: The Data tab shows some information about the distribution

- **Abscissa (X-axis):**
 - **Intervals:** This is the total number of intervals of data represented on the curve.
 - **Maximum:** This is the maximum value found within the time series.
 - **Minimum:** This is the minimum value found within the time series.
 - **Units:** This is the units of measure used for the data in the time series.
- **Ordinate (Y-axis):**
 - **Maximum:** This is the probability of the most probable value in the time series
 - **Minimum:** This is the probability of the least probable value in the time series.
 - **Units:** Units of probability are dimensionless. This value is always defined as *probability*.
 - **Mean:** This is the mean value over the time series.
 - **Std. Dev.:** This is the standard deviation from the mean over the time series.
- **Percentile Calculation:** This section allows you to determine the value below which a given percentage of the distribution lies. Enter a percentage in the box and click **Apply** to display the value. The percentage must be between 0 and 100%, and is limited to two decimal places.

1.6.7 Create Vector Field

Two scalar rectangular grids (*.r2s) or triangular meshes (*.t3s) can be combined to create a vector rectangular grid (*.r2v) or a vector triangular mesh (*.t3v).

To create a vector grid or mesh:

1. Select the grid or mesh object that represents the U-component from the WorkSpace.

2. Select **Tools**→**Create Vector Field...**

The following dialog will appear:

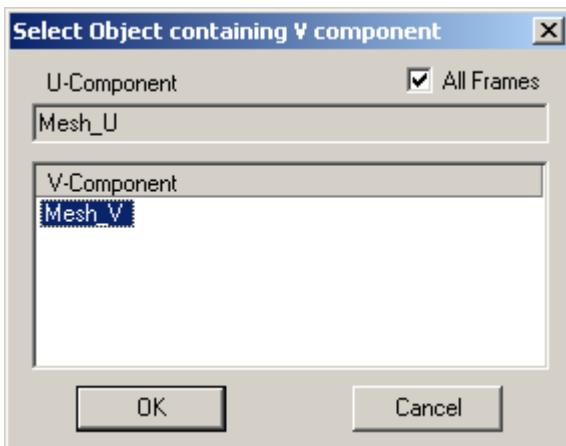


Figure 1.89: The Create Vector Field dialog

The Create Vector Field dialog parameters are described below:

- **U-Component:** The name of the grid or mesh that was selected as the U-Component data object.
- **V-Component:** The names of all data objects found in the workspace that have the same type and spatial geometry as that of the U-Component object.
- **All Frames:** If the V-Component object has the same number of frames as the U-Component object, the All Frames check box will be enabled. Checking this box causes all frames of the scalar objects to be used in the creation of the Vector object. If unchecked, the current frames of each scalar object will be used.

By clicking on the **OK** button, the vector grid or mesh object will be created and added to the WorkSpace.

1.6.8 Mapping Objects

The *Map Objects* tool maps the values (scalar and vector) or data attributes from one object to another object. Values can be mapped from objects that are in the form of a grid or mesh, or that enclose an area (such as closed polylines or polygons). Polygons and points can be used to map values to grids or meshes. Mapping data from a mesh or rectangular grid to a line creates a 3D Line. Data can be mapped to objects such as triangular meshes, rectangular grids, lines or polylines, or point sets. The values will be mapped from one object to the other where there is an overlap.

For example:

- All nodes of a grid or mesh located within a closed line will be given the value of the closed line. Note that only one value can be applied within a polygon.

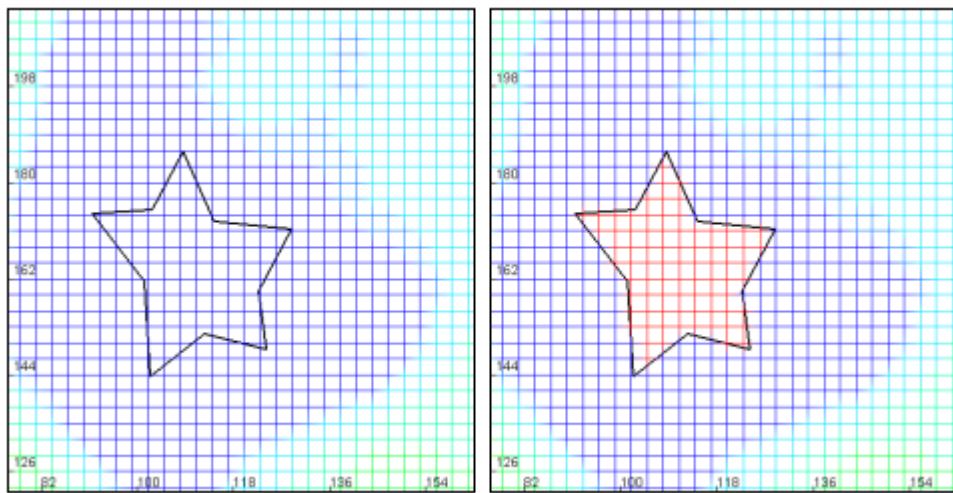


Figure 1.90: The nodes within the polygon have all been given a specific value

- The values from a grid (or mesh) can be mapped to a point set. In the figure on the left, the grid has not yet been mapped to the points. In this example, the points all have the same value. In the figure at the right, the values from the grid have been mapped to the points by spatially interpolating the values on the grid at the location of each point.

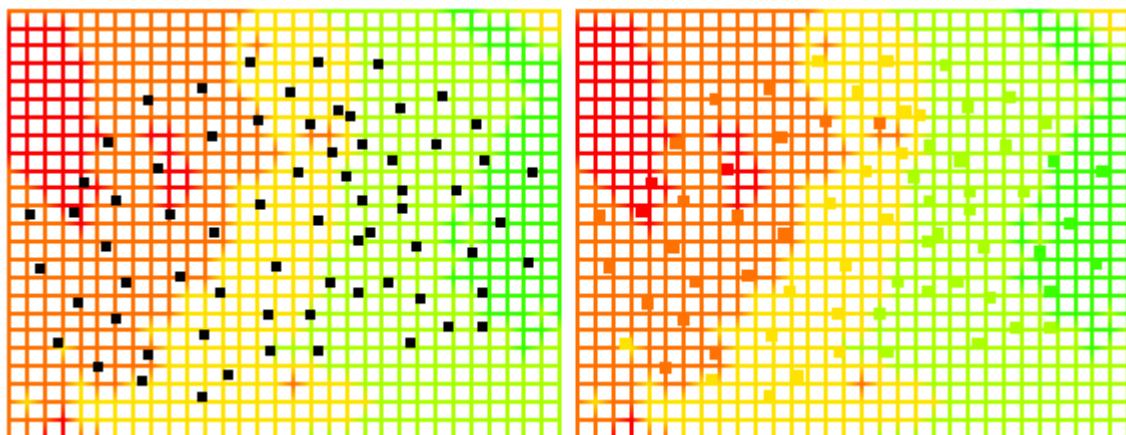


Figure 1.91: The points have been given values interpolated from the data of the grid

To map objects:

1. Open the two data items involved in the mapping. One will be the source object. The other will be the receiving object. The values will be mapped from the source object to the receiving object.
2. Select the receiving object (the object whose values are to be changed) in the WorkSpace.
3. Select **Tools→Map Object**.
4. A dialog will appear, providing a list of all open data items that can be mapped to the receiving object. Select the desired source object.

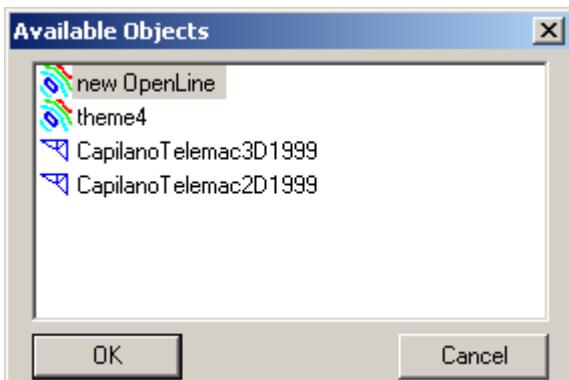


Figure 1.92: This dialog is used to select a source object when mapping

Other possible examples of mapping objects include:

- Vector objects mapped to other vector objects.
- Vector objects mapped to scalar objects. For example, mapping a *.t3v (vector triangular mesh) to an *.r2s (a rectangular grid) object will generate an *.r2v, a vector rectangular grid, showing the interpolated vector values from the mesh at each point on the grid.

1.6.9 Calculators

1.6.9.1 The Calculator for Data Items

The calculator tool performs basic arithmetic operations, such as addition, subtraction, multiplication, or division, on all non-gridded data items. The calculator changes all the values of the data item by the same value or factor. Only one operation may be applied at a time. A possible application of this tool might be to change the units in a data item from metres to feet by multiplying all measurements by 3.280839895.

To use the calculator:

1. Select the data item in the WorkSpace upon which the arithmetic operation will be performed.
2. Select **Tools**→**Calculator**. A dialog will open:

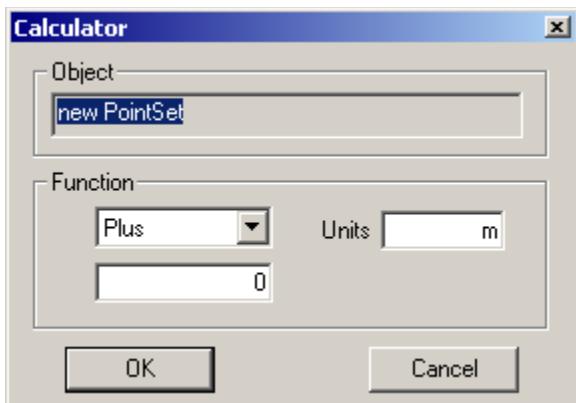


Figure 1.93: The calculator applies an arithmetic operation to a data item

3. In the Function box, select the operation you would like to carry out (**Plus**, **Subtract**, **Times**, **Divide by**) from the pull down menu. In the field beneath this menu, enter the number, factor, or divisor by which the values of the data item should be changed. The units for the data attribute can also be entered at this time.

Note: The units entered are stored in the EnSim file header for the data item, with the AttributeUnits keyword. They may be displayed on the Data tab of an object's property dialog, or in a popup window. Note also that changing the units of an object will not perform a conversion; this entry is for reference only.

If a data item cannot be modified with a calculator used with the calculator, the icon on the tool bar and the menu option in the **Tools** menu will be unavailable.

1.6.9.2 The Calculator for Gridded Objects

A more sophisticated tool is available for gridded data. Both scalar and vector, or static and time-varying gridded data items can be manipulated with this tool.

To use the calculator:

1. Select a gridded data item in the WorkSpace. This is an important step as only data item with the same geometry as the selected object will be available in the calculator.
2. Choose **Tools**→**Calculator**. A dialog window will open:

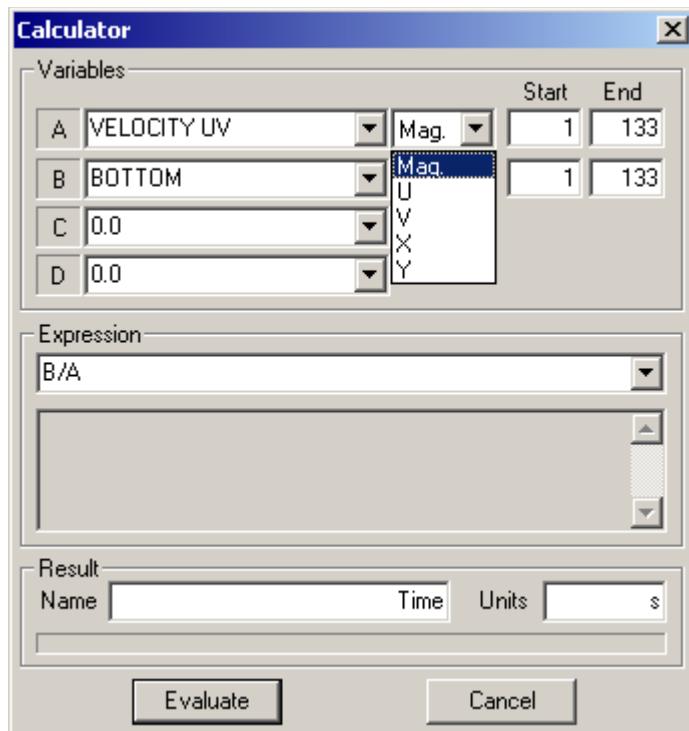


Figure 1.94: The calculator for gridded objects is considerably more complex

- **Variable:** Use this box to assign variable names (A, B, C, or D) to data items currently in the WorkSpace. These variable names are then used to form the equation in the Expression box. In the list box next to the variable name, a list of the available objects which match the geometry of the selected WorkSpace will appear.

For scalar data items, you can choose the range of frames to which the operation will apply. The range (first frame to last frame) is set in the **Start** and **End** boxes. It is important to note that for a range setting larger than one frame, the maximum range of frames selected is used as the template for all other ranges. If a range is later selected that is less than the maximum range, an error message will appear. A range of one frame can be used. If a scalar data item is selected, a list box will appear. The options available in the list box for a scalar object are **Value**, **X**, and **Y**.

For vector data items, you can choose the range of frames in which the operation will apply. The range (first frame to last frame) is set in the **Start** and **End** boxes. If a vector data item is selected, a list box will appear. The options available in the list box for a vector item are **Mag.**, **U**, **V**, **X**, and **Y**. If **Mag.** is chosen, the magnitude of the selected vector data item will be used in the expression. If **U** is chosen, the U component of the selected data item will be used in the expression. If **V** is chosen, the V component of the selected data item will be used in the expression.

- **Expression:** This box allows you to enter a mathematical expression to calculate a new data item. This can include the data items chosen in the **Variable** boxes as well as constants. The mathematical operators available are listed in "The Calculator Expressions", on p. 120.

The **Expression** box also stores previously used expressions from the current session of EnSim.

Once the expression has been evaluated, the parsed expression will appear in the text box below the **Expression** box.

- **Result:** The name of the resulting data item can be entered in the **Name** box and its units may be entered in the **Units** box. Note that changing the units of a data item will not perform a conversion. The **Units** entry is for reference purposes only.

Once the data item's parameters are chosen, and the expression is completed, click on the **Evaluate** button to create the new data item. The new data item will appear in the WorkSpace, and the expression will be displayed in the text box. If the calculation is valid and a name has been provided, the calculator dialog will close. The new data item will be scalar.

If the data item cannot be used with the calculator, the icon on the tool bar and the menu option in the **Tools** menu will be unavailable.

1.6.9.3 The Calculator for Time Series Objects

A calculator is also available for time series. The appearance and functionality of the time series calculator is almost identical to that of the gridded data calculators.

To use the calculator:

1. Select a time series object in the WorkSpace. This is an important step as only time series of the same type (scalar or vector; timestep or explicit times) and same temporal parameters (start time, deltaT, explicit times, etc.) as the selected object will be available in the calculator.
2. Choose **Tools**→**Calculator**. A dialog window will open:

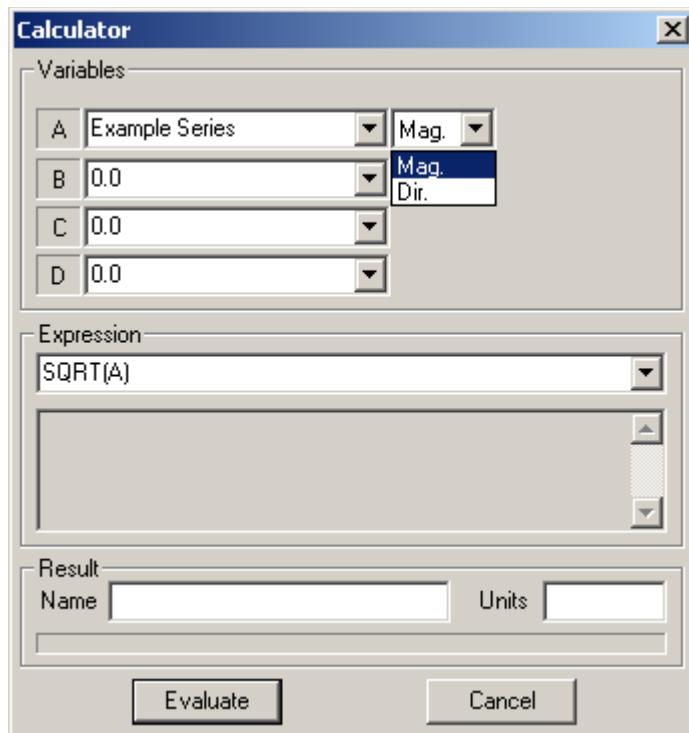


Figure 1.95: The calculator for time series objects

- **Variable:** Use this box to assign variable names (A, B, C, or D) to time series currently in the WorkSpace. These variable names are then used to form the equation in the Expression box. In the list box next to the variable name, a list of the available objects which match the temporal geometry of the selected time series will appear.

For scalar time series, the **Value** component is automatically selected. For vector time series, there are two components to choose from: **Mag.** and **Dir.**. If **Mag.** is chosen, the magnitude or value component of the selected time series will be used in the expression. If **Dir.** is chosen, the direction component of the selected time series will be used in the expression.

- **Expression:** This box allows you to enter a mathematical expression to calculate a new data item. This can include the data items chosen in the **Variable** boxes as well as constants. The mathematical operators available are listed in "The Calculator Expressions", on p. 120.

The **Expression** box also stores previously used expressions from the current session of EnSim.

Once the expression has been evaluated, the parsed expression will appear in the text box below the **Expression** box.

- **Result:** The name of the resulting time series can be entered in the **Name** box and its units may be entered in the **Units** box. Note that changing the units of a time series will not perform a conversion. The **Units** entry is for reference purposes only.

Once the parameters are chosen, and the expression is completed, click on the **Evaluate** button to create the new time series. The new data object will appear in the WorkSpace, and the expression will be displayed in the text box. If the calculation is valid and a name has been provided, the calculator dialog will close.

1.6.9.4 The Calculator Expressions

Both the gridded object calculator and the time series calculator use the same mathematical expression set. The mathematical operators are:

- **+**, **-**, *****, **/**, **^** - arithmetic operators: plus, minus, multiply, divide, exponentiation
- **chs** - change sign (from positive to negative, or vice versa)
- **Pi** - 3.1415926535897932
- **abs** - absolute value
- **sqrt** - square root
- **log10** - logarithm, base 10
- **log** - natural logarithm
- **exp** - exponential (base e)
- **sin** - sine
- **cos** - cosine
- **tan** - tangent
- **asin** - inverse sine (arcsine)
- **acos** - inverse cosine (arccosine)
- **atan** - inverse tangent (arctangent)
- **sinh** - hyperbolic sine
- **cosh** - hyperbolic cosine
- **tanh** - hyperbolic tangent
- **min** - returns the minimum value
- **max** - returns the maximum value
- **mean** - returns the arithmetic mean (average) of all values
- **sigma** - returns the standard deviation of all values
- **()** - parentheses
- **D2R** - convert degrees to radians
- **R2D** - convert radian to degrees
- **>** - (A>B) returns the greater of A and B
- **<** - (A>B) returns the lesser of A and B

For any expression, standard operator precedence applies. Operands for trigonometric functions are in radians.

Examples of valid expressions include:

- A
- 3.45
- A * B - (C / D)
- A * exp(B) - C * log10(D)
- chs(A)
- (pi * A) ^ B
- cos(B) - sinh(A)
- (cos(A / (pi * 180)))
- A < B

1.7 How To - HINTS AND TRICKS

This section is intended to provide you with a quick reference on how to accomplish some useful tasks with EnSim. The items discussed here are based on the fundamental EnSim functions, but may not be readily obvious.

1.7.1 Draping a 3D Image Onto a DEM

EnSim allows the draping of a georeferenced GeoTIFF image over a rectangular grid or triangular mesh for visualization of a 3-dimensional image.

This capability allows you to:

- Locate structures, roads, waterways, or other obstacles that might affect or be affected by the model.
- Observe the limitations of the model.
- More realistically visualize the spatial domain of the model.

To drape an image:

1. Import a GeoTIFF (*.tiff) into the WorkSpace.
2. Create or load a rectangular grid (*.r2s) with the identical spatial extent as that of the image. The resolutions of the grid and the image do not have to be identical. See the sections on "Creating a New Regular Grid" under Creating New Data Items, on p. 70 and "Mapping Objects" under Tools, on p. 113 for more details. Alternatively, load a triangular mesh (*.t3s) with a spatial extent that overlaps that of the image.
3. Drag the rectangular grid or triangular mesh onto the image so that the grid or mesh becomes a child of the image.
4. Drag the image into a 3D view.

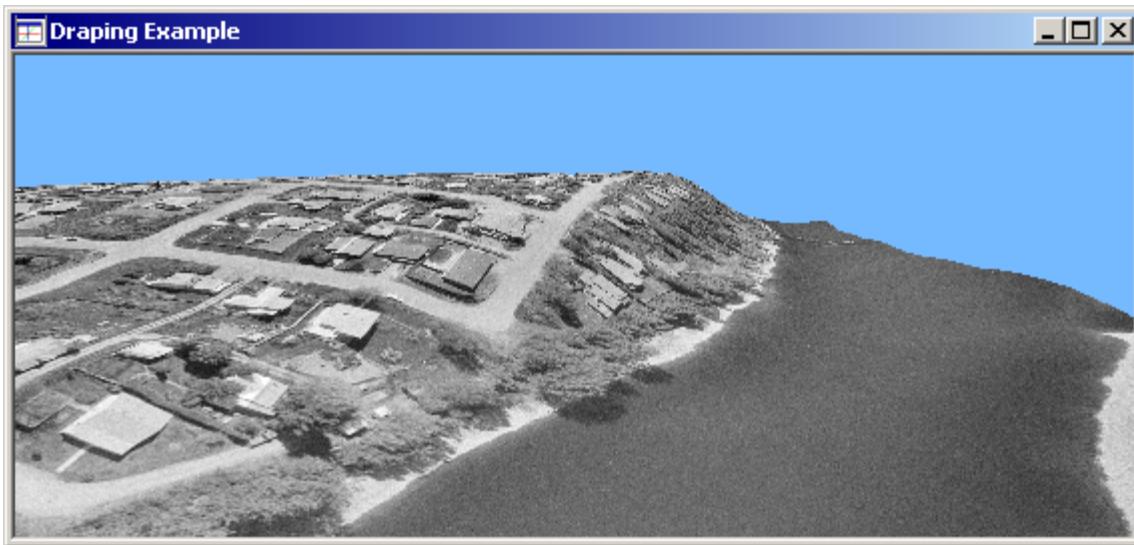


Figure 1.96: This image has been draped onto an elevation model to produce a landscape

1.7.2 Extracting Cross-Sections from Gridded Data

Cross-sections may be extracted from data that is in the form of a grid or mesh.

To create a cross-section or 3D polyline:

1. Display, in a 2D view, the object from which the cross-section will be extracted.
2. Draw a line or polyline (see "Drawing Lines and Closed Polylines" under Creating New Data Items, on p. 69, for more details), or open a file containing a polyline, which defines the xy location of the cross-section. Possible files include *.i2s, *.i3s, *.shp, or *.mif.
3. Select the line object in the WorkSpace.
 - To increase the resolution of the polyline, see the section "Resampling Lines and LineSets" under Selecting Data Items, on p. 80.
4. Select **Tools**→**Map Object**. See "Mapping Objects" under Tools, on p. 113, for more details.
5. Select the grid or mesh from which the cross-section is to be taken and select the  button.

The value applied to each point on the cross-section is interpolated from the surface of the grid or mesh. The cross-section will appear in the WorkSpace as a child of the line, and will have this icon: . The cross-section will have the name of the grid or mesh from which the cross-section was taken, followed by "XSection".



Figure 1.97: This cross-section was taken from a data item named "DEM"

By default, cross-sections appear in the WorkSpace as 3D line objects.

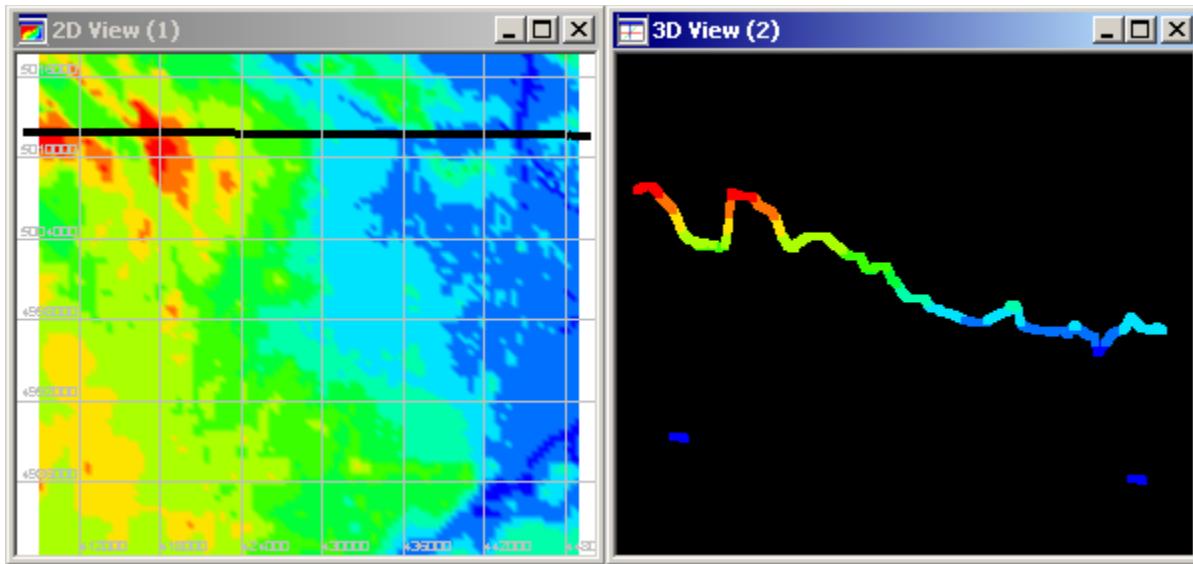


Figure 1.98: The view on the right is the cross-section extracted from the black line in the view on the left

1.7.3 Extracting Cross-Sections from Points and Line Data

Cross-sections may be extracted from non-gridded data (points and lines) by first triangulating this data.

To create a cross-section or 3D polyline from points or line data:

1. Open a line or point set into the WorkSpace.
2. Create a new triangular mesh by using **File→New**.
3. Drag the line or point set onto the mesh.
4. Open the **Properties** dialog and select the button.
5. See section "Extracting Cross-Sections from Gridded Data" under How To - Hints and Tricks, on p. 123, for further steps.

1.7.4 Displaying Two Features of an Object Simultaneously

Sometimes it is desirable to display a spatial object as a surface while retaining the lines of the grid or mesh, or to display an object as filled contours while also showing the distinct isolines that define the contours. It may also be desirable to display two data attributes of an object simultaneously. These things are all done similarly.

What is desired is to display the object in two different display styles, or in two different ways. To do this, the object must be opened in EnSim twice. That is, there must be two of the same object in the WorkSpace. To display an object in "N" different ways, it must appear in the WorkSpace "N" times. Each object is then displayed as one of the desired types.

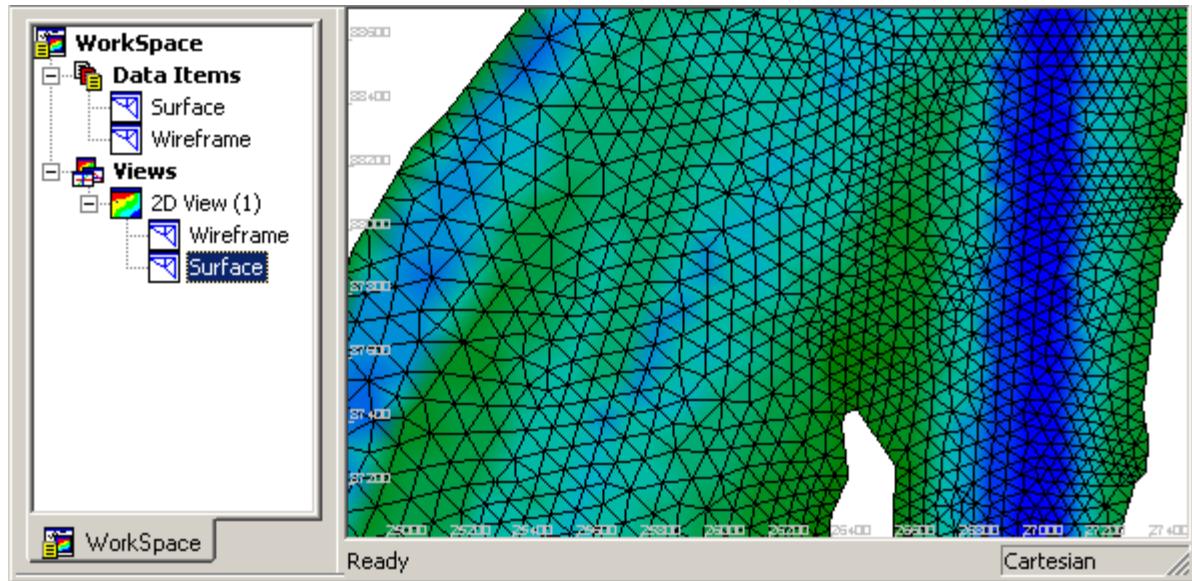


Figure 1.99: This object is being displayed as a coloured surface and a black monochrome mesh simultaneously

If the object is being displayed for editing purposes, be careful to highlight the appropriate object in the WorkSpace to ensure that the correct object is being edited. Also, remember that EnSim considers the objects to be independent of each other. Be careful to edit only one of the objects. Remember that all attributes of an object, not just the one being displayed, can be edited with the **Edit** command in the shortcut menu.

Also, remember the following:

- If you are displaying the objects in a 3D view, ensure that the Scale of both is the same.
- If you are displaying isolines and filled contours, ensure that the colour scale attributes (i.e. min, max, interval, style, levels, etc.) of all objects is the same. Even though an object may be displayed in monochrome, the colour scale defines the values of the isolines.

1.7.5 Displaying Isoline-Outlined Filled Contours

To display an object as filled contours and isolines in the same view, the object need not be opened and displayed twice.

Isolines can be extracted from the grid or mesh, making a new, independent object. To do this, highlight the grid or mesh object in the WorkSpace, select **Tools**→**Extract Isolines**→**Multiple Isolines**. The new line set object will appear in the WorkSpace as a child of the grid or mesh. It can be displayed in the same view as the grid or mesh.

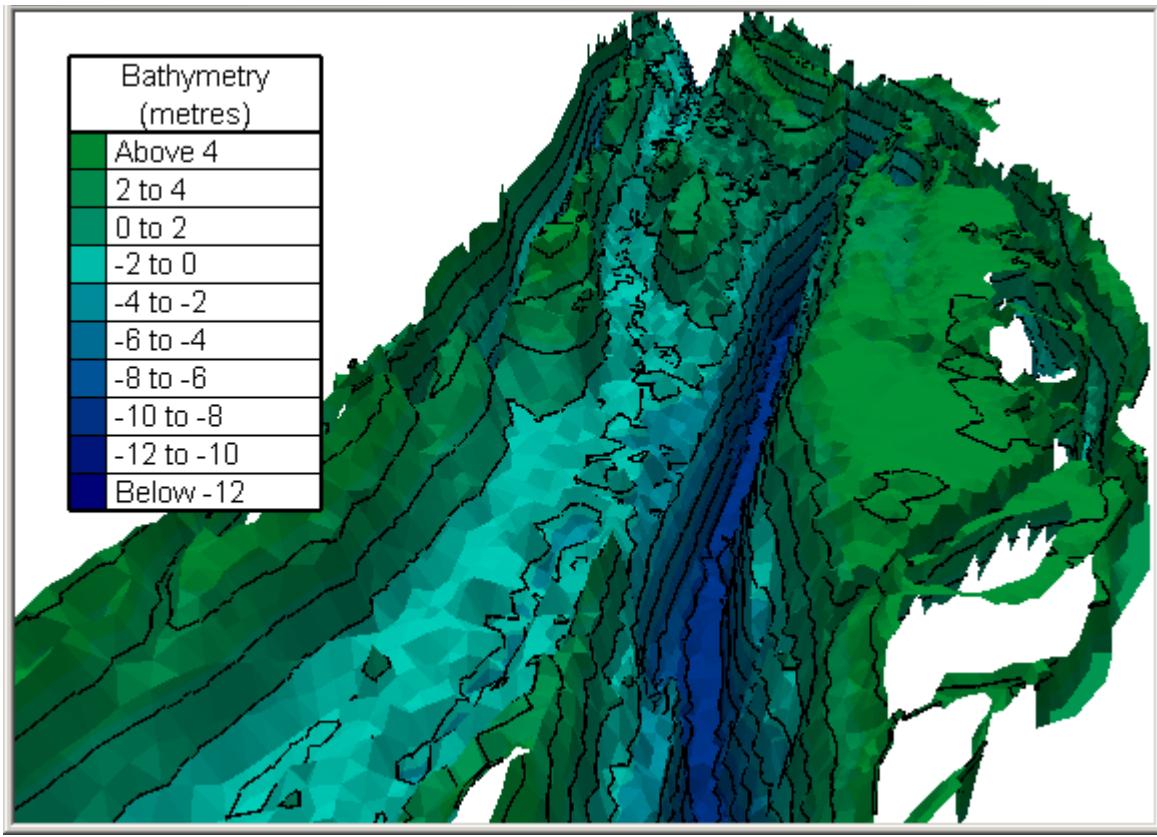


Figure 1.100: A triangular mesh displayed in 3D as a surface of filled contours, with multiple extracted isolines displayed in monochrome black

1.7.6 Creating a Sloping Structure in a Rectangular Grid

When modelling hydrodynamics, sometimes it is desirable to create a simple sloping structure in a regular grid to represent a physical object in the model domain.

To create a sloping structure in a regular grid:

1. Open the regular grid file and display it in a 2D view. Use this grid as a guideline when creating the structure.
2. Create a new point set, or open a saved set, which outlines the shape of the lower and upper bounds of the sloping structure.
3. Edit the points to set their values. Double-click on each point and then right-click to open the shortcut menu. Select **Edit** from the shortcut menu.
4. Create a new Triangulation from the **File** menu. Drag the point set into the new Triangulation object in the WorkSpace. Click the **Triangulate...** button on the Properties dialog of the new Triangulation, which will have opened automatically when the new Triangulation was created. Drag the new Triangulation into a 2D or 3D view to ensure that it is satisfactory in shape.

1. Click on the regular grid object to highlight it in the WorkSpace. Select **Tools**→**Map Object**. Choose the new Triangulation and click **OK**. Display the modified regular grid in a 3D view to see the effect of the addition of the triangulation.

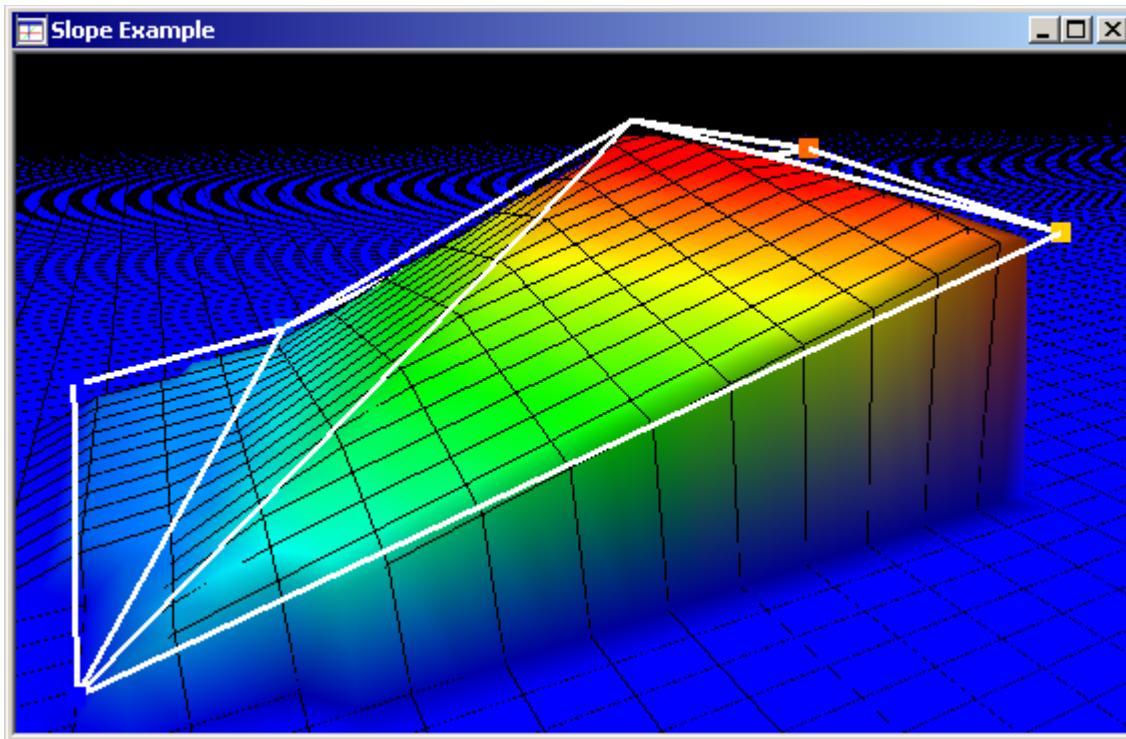


Figure 1.101: A regular grid with a sloping structure, as well as the point set and triangulation used to create the structure, shown in a 3D view

1.7.7 Extracting a Spatial Subset From a Larger Grid

It may be necessary for you to define a spatial subset of a larger grid.

To define a spatial subset from a rectangular grid:

1. Open a 2D rectangular grid into the WorkSpace.
2. Create a new regular grid by selecting **File**→**New**. See "Creating a New Regular Grid" under Creating New Data Items, on p. 70, for more details.
3. Ensure that the spatial domain of the new rectangular grid is within the already-opened rectangular grid.
4. Map the original rectangular grid to the new rectangular grid. See "Mapping Objects" under Tools, on p. 113, for more details.

1.7.8 Extracting a Temporal Subset of Time-Varying Gridded Data

The calculator can be used to extract a temporal subset of time-varying gridded data.

To extract a temporal subset:

1. Open a time-varying gridded data item in the WorkSpace. Ensure that the object is highlighted.
2. Select **Tools→Calculator...**. When the Calculator appears, click the list box for a variable.
3. Select the time-varying data items from the options available. An edit box will appear below **Start** and **End**.
4. Enter the range for the temporal subset. Under **Start**, enter the beginning frame for the subset, and under **End**, enter the last frame for the subset.
5. Enter the variable letter as the expression in the **Expression** box. For example, if you selected the variable "A" in step 3, enter "A" in the **Expression** box. See "The Calculator for Gridded Objects" under Calculators, on p. 116, for more information.
6. Click on the **Evaluate** button to create the new temporal subset. Ensure that the new subset has a name and units.

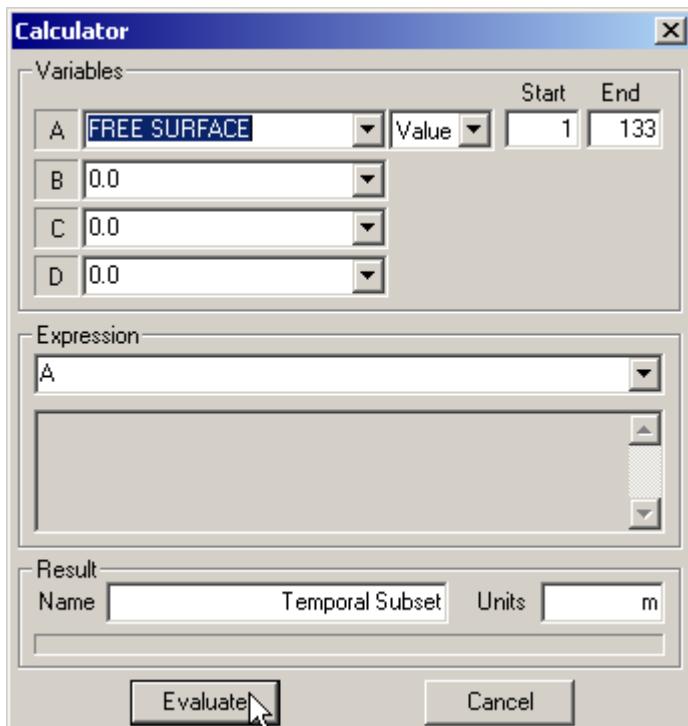


Figure 1.102: This calculator is being used to extract a temporal subset of time-varying data

1.7.9 Digitizing from an Imported Image

Digitizing from a georeferenced image is useful for creating or defining roads, landmarks, or contour lines that may affect a model or help in the visualization of a spatial domain.

To digitize from an imported image:

1. Import a GeoTIFF image into the WorkSpace by selecting **File→Import**.

2. Drag the image into the 2D view.
3. Create a new line set following the path of the viewed object in the image. See "Drawing Lines and Closed Polylines" under Creating New Data Items, on p. 69, for more details. For each line drawn, define its value as something representative of the object viewed. For example, you might define secondary roads as "2" and main roads as "1".
4. Create a new point set. See "Drawing Points" under Creating New Data Items, on p. 68, for more details. For each point set drawn, define its value as something representative of the object. For example, you might define houses as "1" and businesses as "2".

1.7.10 Georeferencing a non-georeferenced GeoTIFF

Non-georeferenced tiffs may be loaded and manually georeferenced.

To georeference a non-georeferenced tiff:

1. Import a GeoTIFF image into the WorkSpace by selecting **File→Import**.
2. Open the **Properties** dialog and select the **Spatial** tab.
3. The pixel size, the coordinates of the origin (or SouthWest corner) of the image can be set and applied.

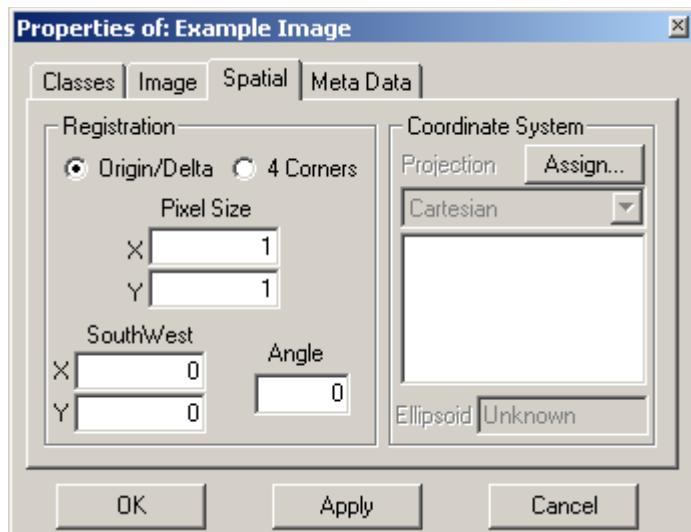


Figure 1.103: Registering a GeoTIFF image at the origin

Alternatively, the image may be stretched into place by setting the coordinates of the four corners. To do this, click on the 4 corners button and set pixel size and the corner coordinates.

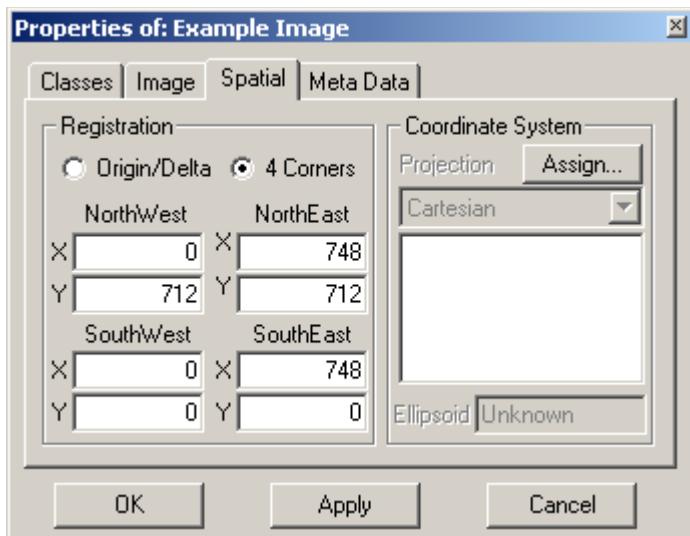


Figure 1.104: Registering a GeoTIFF image at the corners

4. If the coordinate system is known, it should be assigned (see "Coordinate Systems - Assigning Projections", on p. 27).
5. On saving the GeoTIFF image, the origin and pixel size will be written to the file. Only non-rotated images in a rectangular space can be saved. A rotated or stretched image may be viewed only.

1.7.11 Classification of a GeoTIFF Image

The pixels of a GeoTIFF image may be categorized into classes by associating the pixel value to a colour and a class name. In Ensim, this is only possible with 8-bit and 16-bit images.

To classify a GeoTIFF image:

1. Import the GeoTIFF image into the WorkSpace by selecting **File**→**Import**.
2. Open the **Properties** dialog and select the **Classes** tab.

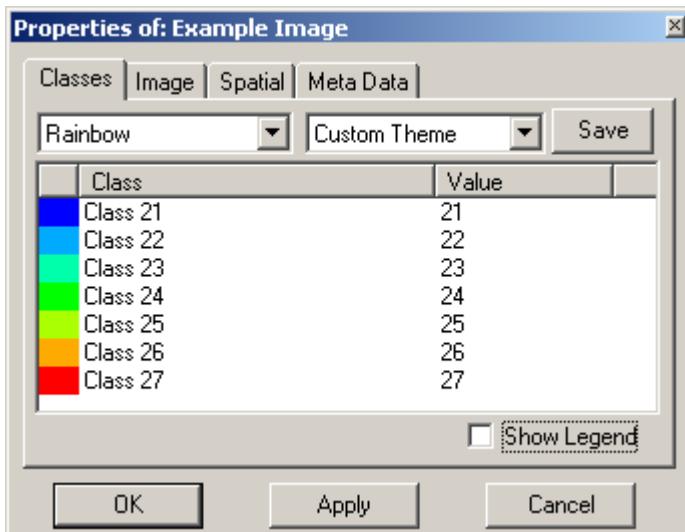


Figure 1.105: Classes tab for a GeoTIFF image

3. The colour value of each image pixel is associated with a class name. In the figure above, the image contains seven distinct values ranging from 21 to 27. The group of class colours, names and values is defined as a theme and may be saved as an ASCII .thm file (see "GeoTIFF Theme files [thm]", on p. 305 for more information). You can create a **Custom Theme** or selecting from a list of predefined saved themes.

To create a Custom Theme:

1. With **Custom Theme** selected, you can select from predefined colour schemes which include **Default Colours**, **Rainbow**, **Grayscale**, **USGS Reduced**, **40 Nice Colours**, **RGB Interpolation**, and **HSV Interpolation**. on the colour or name of an individual class to set their own colours and names.
2. Also within the **Custom Theme** mode, you can edit individual class colours and names by clicking on them.
3. To save the created scheme, press the Save button. This file will be saved to an ASCII *.thm file and stored in the bin\Templates\GEOTIFF directory.

To choose from a predefined theme:

1. All saved theme files found in the bin\Templates\GEOTIFF directory are selectable from the list in the top right of the dialog. Once a predefined theme is chosen, changes to colour schemes, class colours and class names are not allowed.
2. Several documented land cover theme files are provided:
 - **EC** - Environment Canada
 - **LA-GAP** - Louisiana GAP Analysis Project.
 - **USGS NatAtlas_LC** - USGS National Atlas Land Cover
 - **EOSD** - Earth Observation for Sustainable Development of forests

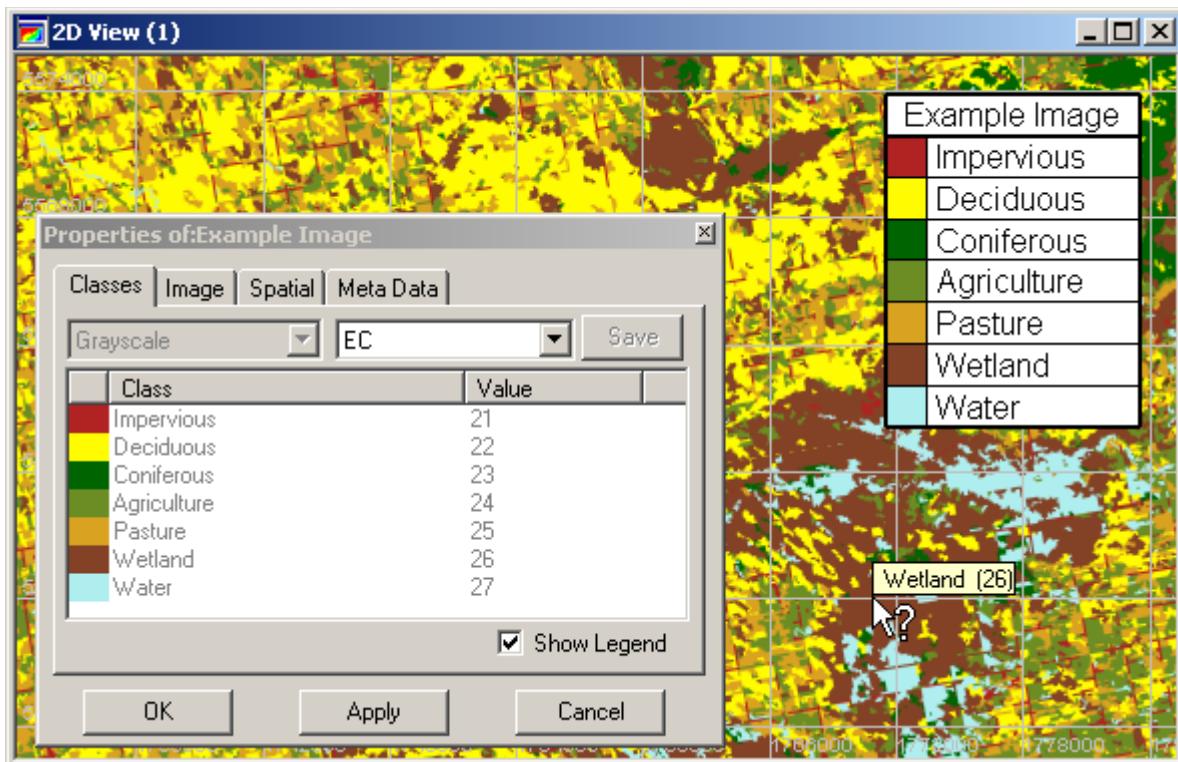


Figure 1.106: Land Classification visualized from a GeoTIFF image

To reclassify a GeoTIFF

1. With the GeoTIFF object selected in the WorkSpace, select **Tools**→**Reclassify Image...** from the menu bar, or right-click and select **Reclassify Image...** from the shortcut menu.

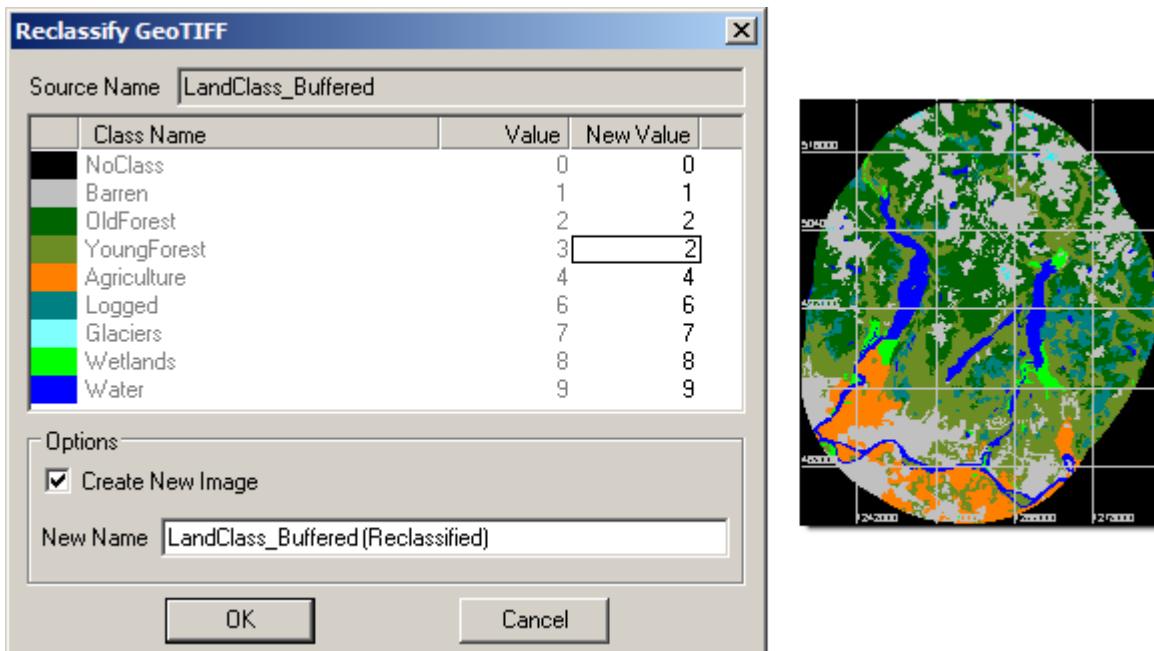


Figure 1.107: Reclassifying a GeoTIFF allows you to combine similar categories. This image has nine categories

2. In the **New Value** column, select each value to be changed and enter the new value.
3. To create a new object containing the changes without modifying the original, leave the **Create New Image** checkbox checked and enter the new object name in the **New Name** box.

To save the changes to the original object, clear the **Create New Image** checkbox.

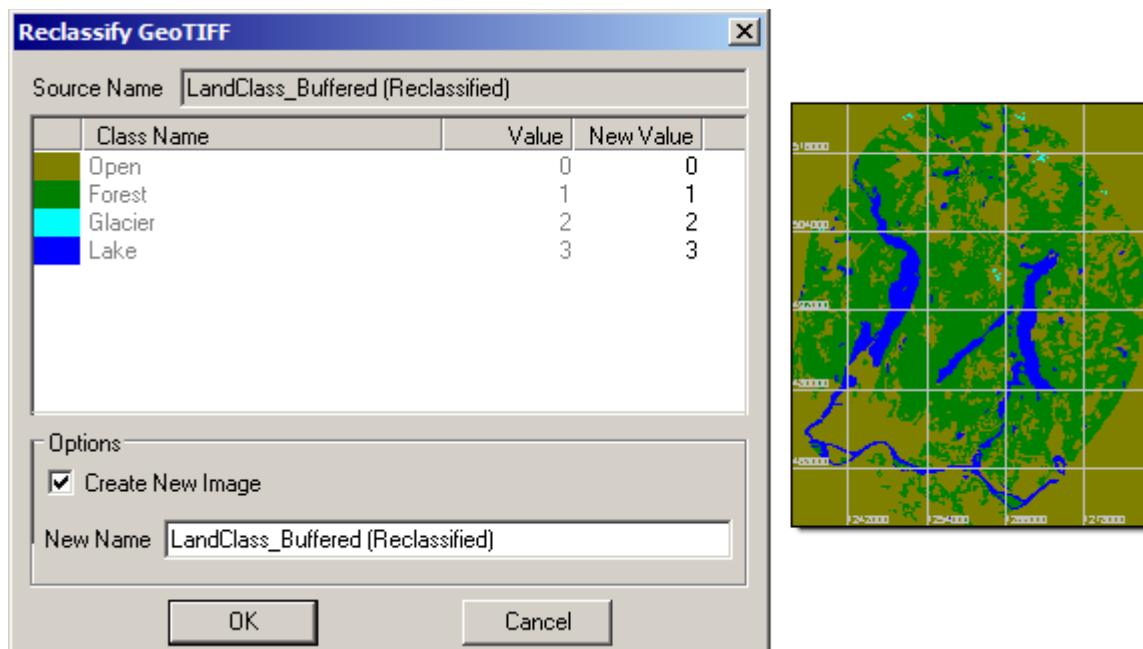


Figure 1.108: The image has been reduced to four categories, suitable for the HBV-EC model

4. Click **OK**.

2 GREEN KENUE

Green Kenuue is an EnSim application that provides an integrated numerical modelling environment for hydrological models, such as WATFLOOD, HBV-EC, or GEN1D. Green Kenuue consists of the core EnSim features common to all EnSim applications, plus additional features that are specific to hydrological modelling. This section describes the interface and tools specific to Green Kenuue.

Green Kenuue allows you to create much of the information required to run a hydrological model by creating a watershed object. From the watershed object, both the map file required by the hydrological model WATFLOOD and the HBV-EC parameter file can be generated. The tools available in Green Kenuue are integrated with general EnSim tools, allowing grids, spatial GIS-type information, and model results to be viewed and analyzed.

2.1 WATERSHED OBJECTS

A *watershed object* is a very important data item in Green Kenuue since it contains the basic geographical and geophysical data necessary to run a hydrologic model.

The watershed object is created in Green Kenuue from a regular (rectangular) grid of georeferenced elevations. This grid can be generated by Green Kenuue from a georeferenced point set, or a digital elevation map (e.g. CDED, DTED, DEM etc.) may be used directly. The elevation data of the grid is used by Green Kenuue to generate the channels, or flow paths, of the water as it travels overland to the watershed outlet, and the boundary of the watershed. See "Creating a New Regular Grid" under Creating New Data Items, on p. 70, for more details on grid generation.

Accordingly, a watershed object contains three object types:

- a processed DEM (digital elevation map) grid in the format of a regular, or rectangular, grid that describes the topography of the land
- a network that describes the path of the flow of surface water through the watershed
- a watershed boundary or basin (there may be more than one basin in a watershed object)

In the WorkSpace, the watershed object will appear similar to the following example:

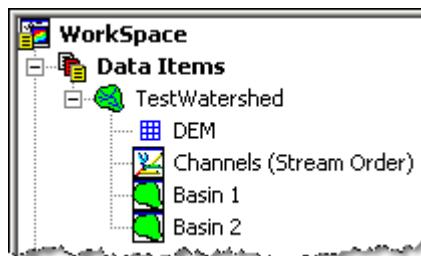


Figure 2.1: An example of a watershed object

2.1.1 Opening an Existing Watershed Object

To open an existing watershed object, select **File→Open** from the menu bar, or click the  button. Watershed objects have the file extension *.wsd. Once opened, the watershed object will be listed under the **Data Items** category of the WorkSpace.

2.1.2 Importing a Watershed from Topaz

Watershed files created with the Topaz software can be imported into Green Kenu to create a watershed object. The Topaz files must be in ArcInfo ASCII grid file (*.arc) format. An ArcInfo grid file from Topaz may be added to the EnSim WorkSpace and displayed in a 2D or 3D view.

To create a new watershed object from Topaz files, select **File→New→Watershed from Topaz files...** from the menu bar. A dialog will appear requesting the paths of the Topaz DEM, watershed boundary, drainage directions and upstream drainage area files.

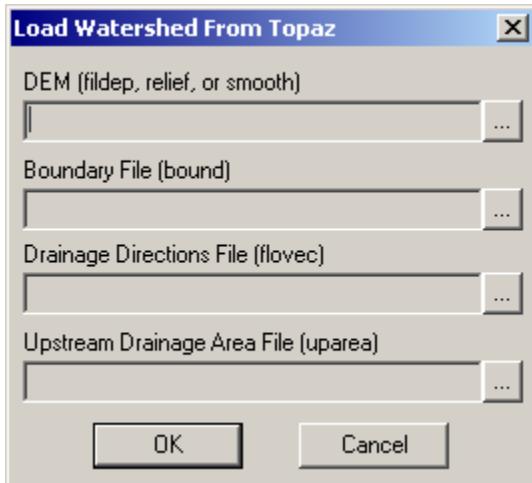


Figure 2.2: This dialog box loads a watershed from Topaz files

Click  to browse for the files. Once a file has been specified, Green Kenu will look within the same directory for the associated files. If any of the file suggestions are incorrect, use the browse button to change any of the automatically specified files.

2.1.3 Creating a New Watershed Object

To create a new watershed object, select **File→New→Watershed...** from the menu bar, or click on the  button on the Tool bar. A watershed object will be created and listed under the Data Items category, with the component files empty.



Figure 2.3: A new watershed contains empty component files

Load a regular grid of elevations or a DEM using the button. The grid file may be in *.r2s format, *.dem format, or *.hgt format. See "Loading and Importing Data Items" under Data Items, on p. 10, for more information. You can load a DEM directly into the watershed by right-clicking on the DEM child object of the New WaterShed object in the WorkSpace and selecting **Load From File...** from the shortcut menu.

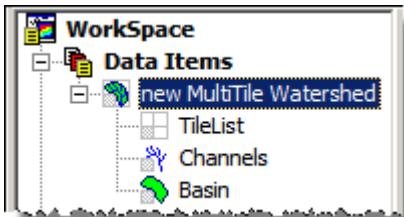


Figure 2.4: These icons indicate that the watershed contains multiple source tiles

You can also create a watershed from multiple tiles. This is called a multi-tile watershed, and can be created by selecting **File→New→Multi-Tile Watershed** from the menu bar. Multi-tile watersheds can contain *.r2s or *.dem source data, and use slightly different icons. In addition, the watershed object cannot be displayed in a View; if the TileList is moved to a View, only the tiles' outlines and filenames will be shown.

To create a new grid from another data type, use the grid generation tools described in the section "Creating a New Regular Grid" under Creating New Data Items, on p. 70. The elevation grid (DEM) is the most important feature in the watershed object, as all other components—the channels and the basin outline—are based on the information it contains. Care should be taken to create a quality grid.

Drag the grid file onto the watershed object's empty DEM component, within the workspace. Select a flow algorithm and press the generate button. Green Kenuer will create a default channel network and a default basin.

2.1.3.1 Watersheds

The watershed object is the parent object for the three children. The watershed properties tab describes the methods for watershed delineation and its associated component objects.

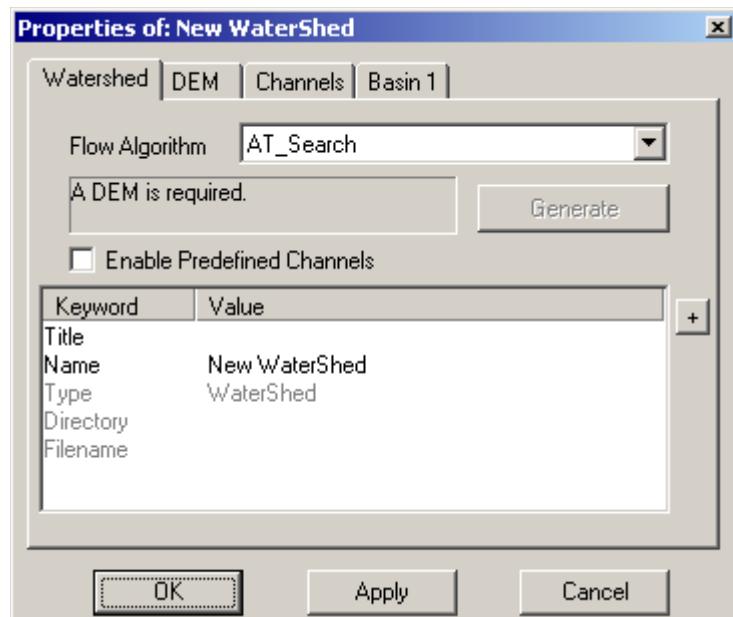


Figure 2.5: This newly created watershed object does not yet contain a DEM

2.1.3.1.1 Flow Algorithms

There are two algorithms available for delineating a watershed from a Digital Elevation Model (DEM):

- A^t algorithm (based on an algorithm used by Ehlschlaeger at the U.S. Army Construction Engineering Research Lab).

The A^t algorithm is a tree search algorithm. The A^t algorithm does not modify the DEM, which allows for more genuine channel delineation. This algorithm is iterative and does not have to deal with deep recursion and the subsequent memory problems associated with large DEMs.

- Depressionless DEM algorithm (developed by Susan Jenson).

The Jenson depressionless DEM algorithm delineates a watershed by lifting nodes within the DEM to remove all depressions. A depressionless DEM allows for simple channel delineation as there is a down or zero slope at every node.

This algorithm has been implemented recursively, which may lead to memory problems with very large DEMs.

The text box located below the flow algorithm options briefly describes the flow algorithm chosen. If no DEM is linked to the watershed, the box will inform you that a DEM is required.

2.1.3.1.2 Delineating a Watershed

To delineate a watershed:

1. Select the desired flow algorithm from the **Flow Algorithm** list box.

2. Select the **Generate** button. Information on the modifications to the watershed and progress of the watershed delineation appear in the following dialog.

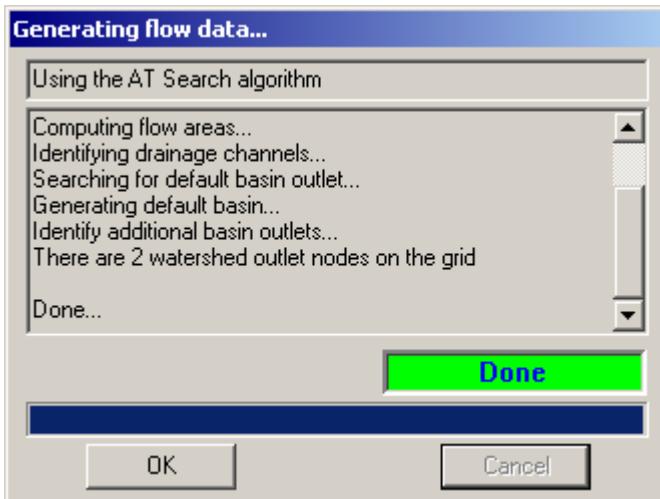


Figure 2.6: This dialog shows the progress of a watershed being generated

Once this dialog has been accepted by clicking **OK**, two new objects, Channels and Basin 1, will appear in the WorkSpace as children of the watershed object.

2.1.3.1.3 Delineating a Multi-Tile Watershed

To delineate a multi-tile watershed:

1. On the **Properties of: new MultiTile Watershed** dialog, use the **Add** button to add the tiles that describe the area of the watershed.
2. Once all of the relevant DEMs have been added, click **Delineate WaterShed** to create the Channels and Basin objects.

The delineation method used by the Multi-Tile Watershed object is a modified version of the A^t Search algorithm.

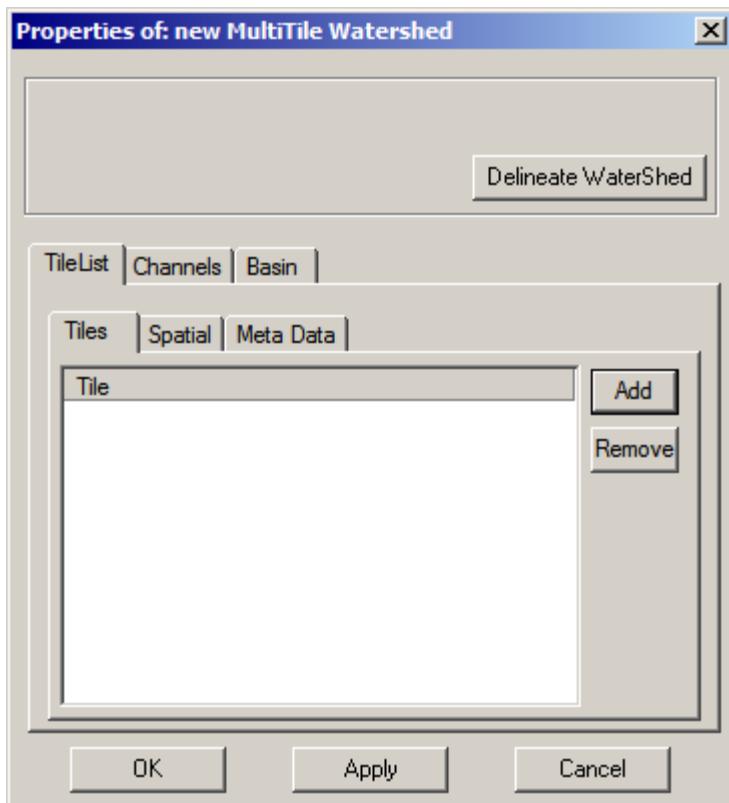


Figure 2.7: This dialog creates a watershed from multiple tiles

2.1.3.2 DEMs

The **DEM** (digital elevation model) tab from the watershed properties dialog describes the rectangular grid (*.r2s) containing the digital elevation data used in the delineation of the watershed. The **Meta Data** tab indicates the source data from which the DEM was created. The DEM appears as a child of the watershed object in the WorkSpace.

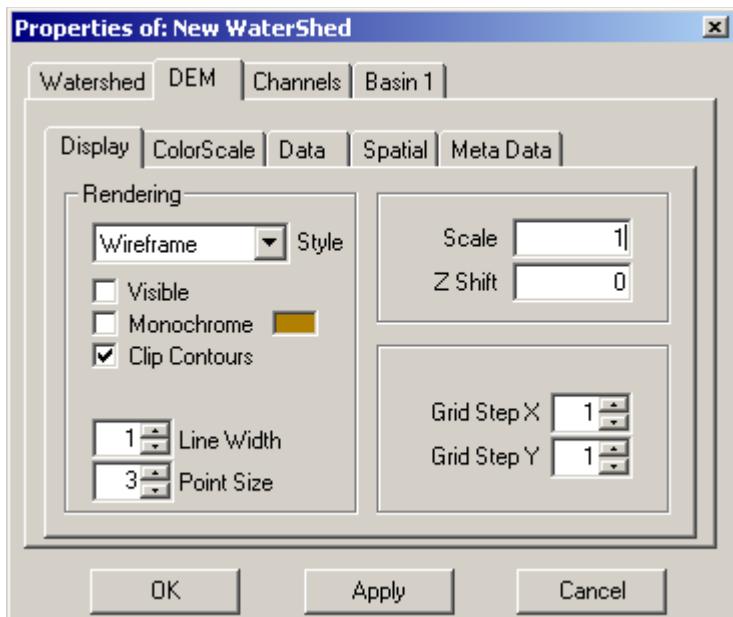


Figure 2.8: The DEM tab is accessible as a child of the watershed object

2.1.3.2.1 Checking for Errors and Editing the DEM

Delineation of channels and the watershed boundary by EnSim is only as good as the DEM. It is important to check the channels and watershed boundary with existing data (for example, GIS data) to ensure correct location of streams and watersheds. GIS data images of the actual paths of streams and rivers can be imported into Green Kenu and displayed in a view along with the channels generated by EnSim.

DEM's with low resolution may introduce significant errors during the removal of depressions and flat areas.

For these reasons, it is a good idea to examine the DEM closely.

For significant errors, edit the DEM (ideally, based on accurate field data), and regenerate the watershed object. After the DEM has been edited, create a new watershed object. This will ensure that the Channels and the Basin are compatible with the edited DEM. For information on how to edit a grid, see the section "Selecting Data Items" under Tools, on p. 73.

2.1.3.3 Channels and Flow Paths

The *channels* or *flow paths* of the watershed object delineate the drainage path of surface water through the watershed. In nature, channels are only formed if water flows over the land's surface for a relatively long distance. Since Green Kenu refers to all paths of the flow of surface water through the watershed as "channels," regardless of their upstream drainage area, the term "channels" does not necessarily refer to the delineation of existing streams, rivers or waterways.

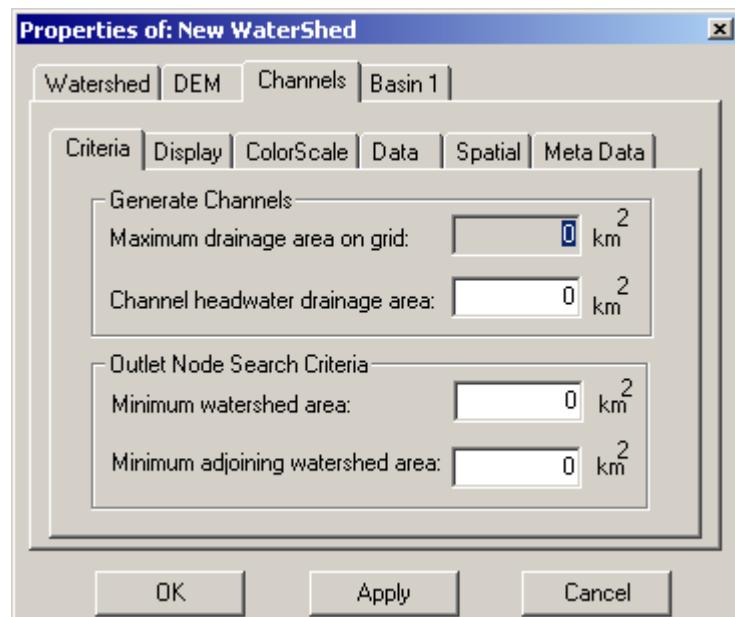


Figure 2.9: The Channels tab is found on the watershed Properties dialog

Flow paths are represented by network segments and may be saved in ***.n3s** format (see "Networks [n3s]", on p. 296, for more information on file formats).

2.1.3.3.1 Channel Attributes

Green Kenu defines the channels of the watershed by connecting the nodes of the DEM that fall along the path of surface water flow. Each channel, or flow path, has a stream order and a drainage area. Stream order and drainage area are data attributes of a channel and can be found under the **Data** tab of the Channel object's **Properties** dialog, or can be displayed in a popup window. See "Properties of Data Items" under Data Items, on p. 17, and "Data Probes" under Probing Data, on p. 83, for more information.

Stream order (or Strahler order) is a measure of the relative size of a channel. Headwater channels have an order of one. When two channels of the same order meet, the single downstream channel that is formed has an order that is one greater than its two upstream channels.

The drainage area of a channel is the land area upstream over which surface water drains to the most downstream node of a segment of a channel. A channel segment is that part of a channel that lies between two of the channel's tributaries. The most downstream node of a channel segment is its watershed outlet node. See "Watershed or Basin Outlet Nodes", on p. 147, for more information. Since the channels are just lines delineating the flow path of water along the DEM, and data attributes of channels are based on the information contained in the DEM, information regarding the drainage area can be extracted from the DEM. See "Extracting Drainage Area" under Hydrologic Tools, on p. 152, for more information.

2.1.3.3.2 Displaying Channels

Channels can be viewed by dragging the Channels object into a 2D or 3D view. The colour scale of the flow paths describes either stream order or drainage area. The **Data** tab of the Channels **Properties** dialog allows you to select the attribute being displayed, as indicated by a green check mark.

In addition to the flow path, the Channels object can optionally display watershed outlet nodes. A circle around a node identifies a watershed outlet node that meets the outlet node search criteria parameters. See the section "Watershed or Basin Outlet Nodes", on p. 147, for more information. Watershed outlet nodes are not marked in the figure below. The green line is the watershed boundary.

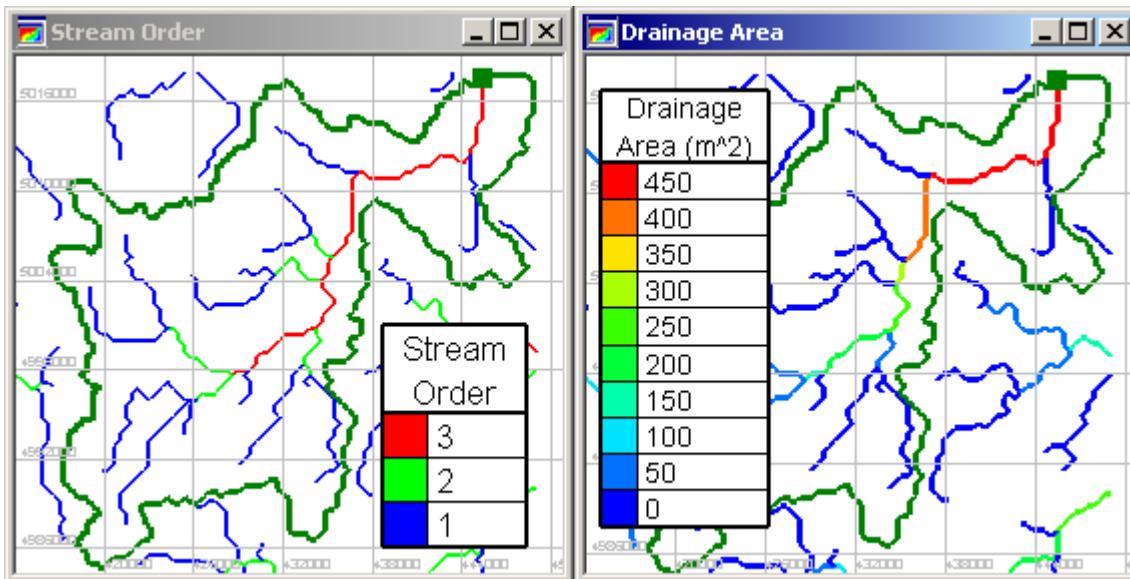


Figure 2.10: These examples indicate the stream order and drainage areas of the watershed bounded by the green line

The extent of definition of the channels, determining whether minor channels will be displayed, can be adjusted in the **Criteria** tab of the Channel's **Properties** dialog.

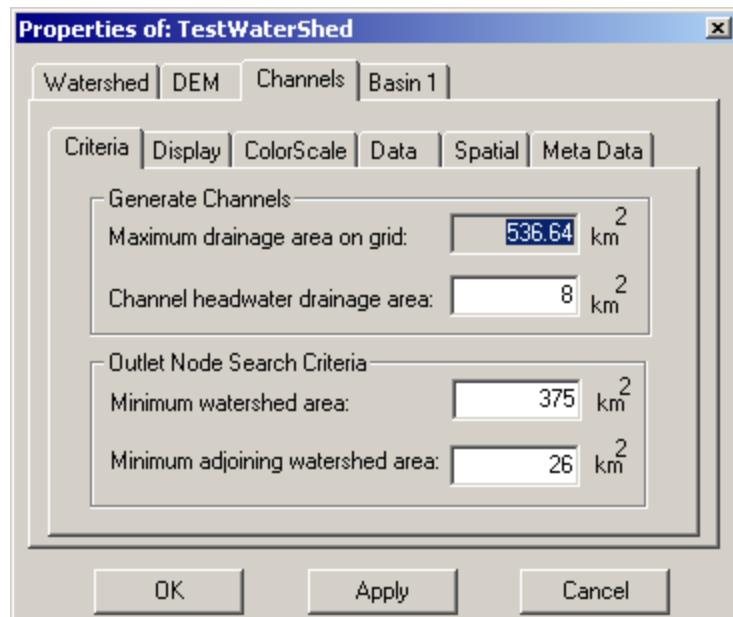


Figure 2.11: The Criteria tab determines which channels and outlets are displayed

The extent to which flow paths are displayed depends on the parameter **Channel headwater drainage area**. This parameter is the area upstream of a particular channel. Only channels that conduct flow from areas greater than the specified channel headwater drainage area will be displayed. These results will also be used in calculating the number of channels in each cell, a parameter required by the Watflood Map file.

To help you with the selection of this drainage area parameter, the maximum drainage area of the grid is displayed as a non-editable parameter, **Maximum drainage area on grid**. As well, the drainage areas for each node of the watershed can be extracted by selecting the DEM object in the WorkSpace and selecting **Tools**→**Watershed**→**Drainage Areas** from the menu bar. The drainage area grid object can be viewed in a 2D or 3D view.

To view more or fewer channels:

1. Ensure that the Channels object is being displayed in a view.
2. Double-click the Channels object. The **Properties** dialog will appear. Select the **Criteria** tab.
3. In the **Generate Channels** area, adjust the parameter **Channel headwater drainage area**. Increasing this parameter will decrease the number of flow paths displayed, and accordingly decreasing this parameter will increase the number of flow paths displayed.

Note: Very small channel headwater drainage area values may slow down channel regeneration.

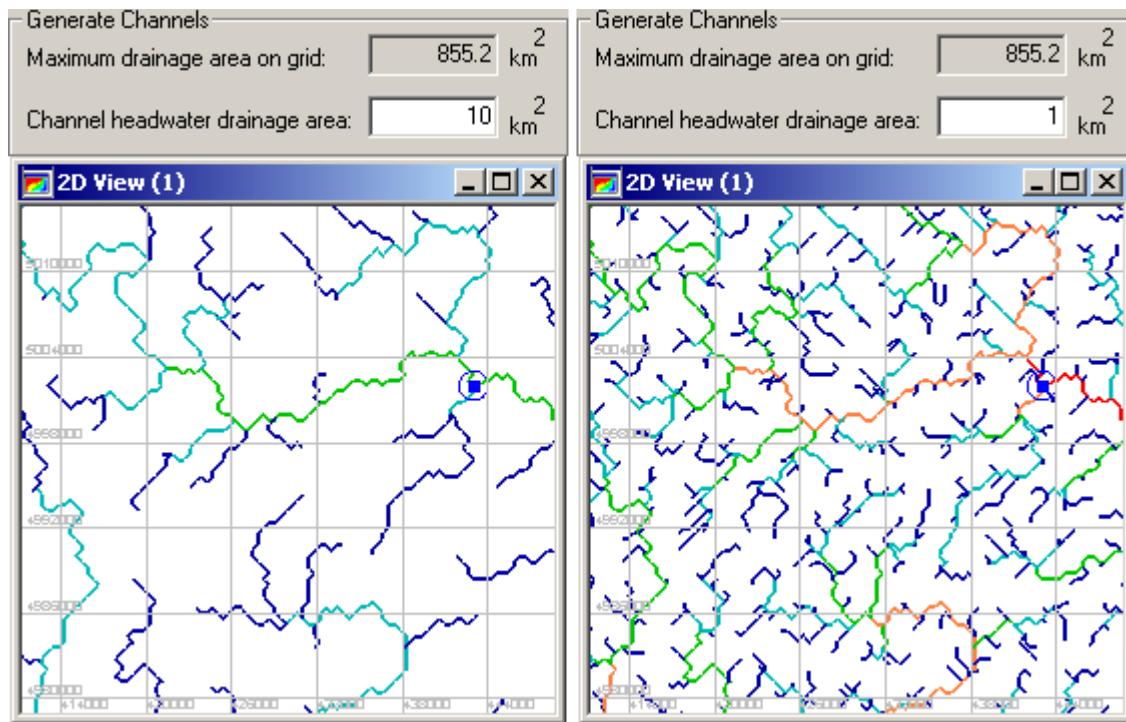


Figure 2.12: As the drainage area decreases, more channels are displayed within the watershed

2.1.3.3.3 Editing the Channels

Before adjusting the channels, their flow paths should be checked against "real" channels. Import GIS data into EnSim that contains information about the rivers and streams in the watershed that is being modelled. See "Supported Foreign File Formats [EnSim Core]", on p. 304, for information about compatible file formats. The channels should be adjusted if they differ significantly from the channels in the GIS data.

Since the channels are generated based on the DEM, the DEM must be edited to alter the channels. The DEM should be modified in such a way as to cause the surface water to flow along the proper path. Remembering that water flows downhill, the elevation at nodes of the DEM along the "correct" flow path of surface water must be lowered to encourage water to flow in that direction. Also, nodes of the DEM can be raised to prevent water from flowing in a certain direction. Take care when altering areas of the DEM that are within the watershed boundary. Usually the elevation does not need to be increased or decreased by much to redirect the flow path of water.

See "Checking for Errors and Editing the DEM", on p. 141, and "Selecting Data Items" under Tools, on p. 73, for more information.

2.1.3.3.4 Using Predefined Channels

Although the channel-delineation algorithms used by Green Kenu are powerful, there may be cases in which the predicted channel paths differ from the observed placement of the channels, or simply watersheds for which you have accurate data available showing the path of an

existing channel. In these cases, it's possible to use the **Predefined Channels** tab to include information about actual channel paths in the watershed object.

Note: This option is not available when working with a Multi-Tile Watershed object.

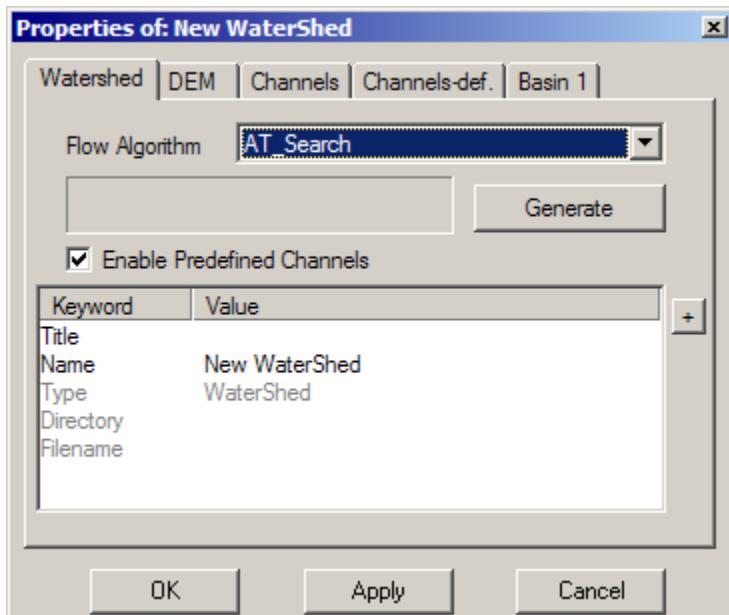


Figure 2.13: The *Enable Predefined Channels* box makes the *Channels-def.* tab available

The Predefined Channels tab (abbreviated as **Channels-def.** within the dialog) lists one or more lines that have been added to the Watershed object in the form of a 2D Line set object.

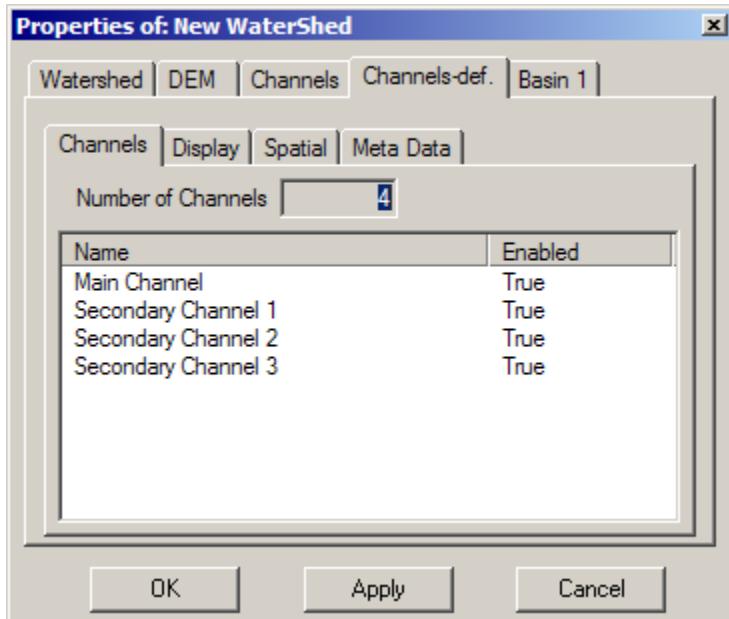


Figure 2.14: This dialog tab lists each of the predefined channels

Once a channel has been added to this tab, it cannot be removed, although it can be disabled by clicking on the **True** or **False** descriptor in the **Enabled** column and clicking **Apply**.

You can deactivate the **Channels-def.** tab by selecting the **Channels-def.** object in the WorkSpace and hitting <Delete>, or by selecting **Remove** from the object's shortcut menu.

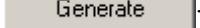
To add a predefined channel to a Watershed:

1. To add a predefined channel to a watershed object, the channel must be loaded as a 2D or 3D Line Set object.

If your channel data isn't in a form that EnSim can access, you can enter the data manually by creating an Open Line object. See "Drawing Lines and Closed Polyline", on p. 69 for more information on this process.

2. Once you have a 2D or 3D Line Set object with your data open in the WorkSpace, there are two ways to create the **Channels-def.** child object of the Watershed.
 - Click the **Enable Predefined Channels** box on the Watershed Properties dialog. See Figure 2.13 on p. 146 for details.
 - Drag the 2D or 3D Line Set object to the Watershed object.
3. To add additional channels, drag the Line Set object describing the channels to the **Channels-def.** object.

The **Number of Channels** value on the **Channels-def** tab of the Watershed properties dialog will increase by one for each set of channels added, and the new channels will be listed and enabled.

4. Go back to the Watershed tab of the properties dialog, change the **Flow Algorithm** to **AT_Search**, if it isn't already, and click .

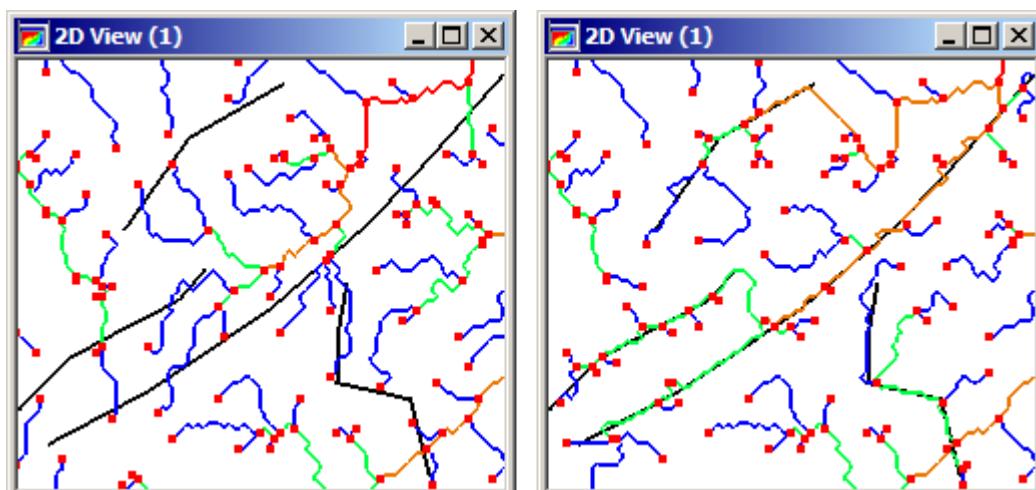


Figure 2.15: These channel objects are shown before (left) and after (right) the predefined channels (black) were added.

2.1.3.3.5 Watershed or Basin Outlet Nodes

A *watershed outlet node* marks the drainage outlet of a basin that meets user-defined criteria. This node, if present, occurs at a point on the channel just before the channel joins with one or more other channels.

On the **Criteria** tab of the Channels **Properties** dialog, watershed outlet nodes are defined by two parameters:

- **Minimum Watershed Area:** All outlet nodes with an upstream drainage area greater than this value will be potential watershed outlets.
- **Minimum Adjoining Watershed Area:** Watersheds adjacent to the potential watershed outlets must have upstream drainage areas greater than this value.

To display more watershed outlet nodes, decrease these parameters. This will cause smaller drainage areas to qualify as watersheds. To display fewer watershed outlet nodes, increase the parameters.

The display of watershed outlet nodes is controlled by the **Target Outlets Visible** checkbox, in the **Display** tab of the Channels object's **Properties** dialog.

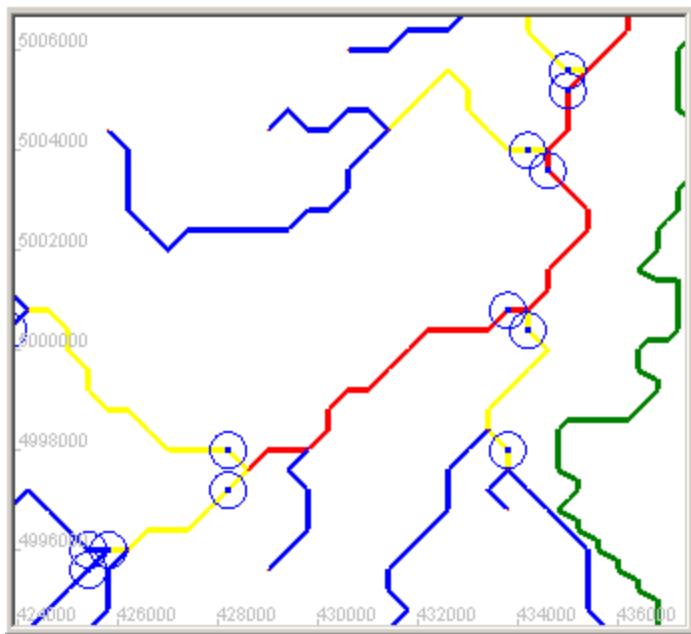


Figure 2.16: Watershed outlet nodes are indicated by circles on this image

To select an outlet node:

1. Double-click anywhere within the watershed outlet node's circle.

To select a channel node near a watershed outlet node:

1. On the **Display** tab of the Channels object's **Properties** dialog, uncheck the **Target Outlets Visible** box. The circles surrounding the outlet nodes will disappear.
2. Double-click on the channel node. Channel nodes are located at the endpoints of each channel line segment.

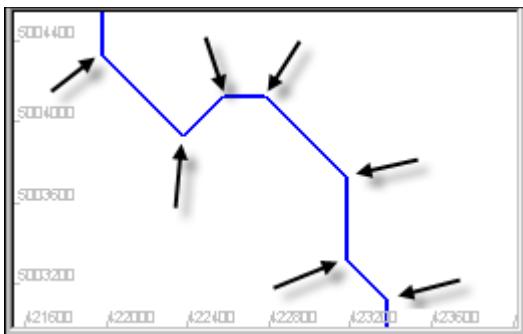


Figure 2.17: The arrows indicate nodes along this channel segment

2.1.3.4 Basin or Watershed Boundaries

The *watershed boundary* is an isoline that defines the watershed. The watershed or *basin* consists of all the nodes of the DEM that drain towards a watershed outlet node. Watershed boundaries can be saved independently of the watershed object.

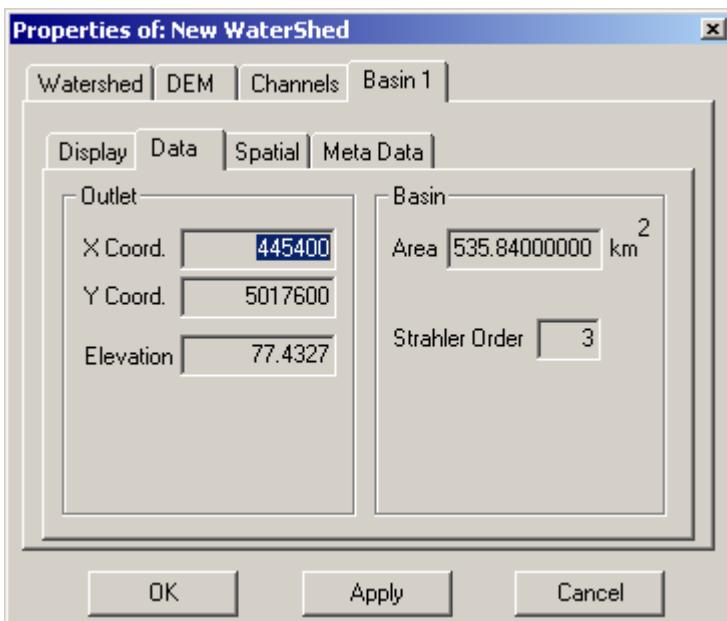


Figure 2.18: The properties of a basin object cannot be directly edited

The following properties, visible on the **Data** tab as read-only fields, cannot be edited without changing the watershed object:

- **Outlet** **area**:
 - **X Coord**: x coordinate of the outlet node
 - **Y Coord**: y coordinate of the outlet node
 - **Elevation**: elevation of the outlet node, in m
- **Basin**:
 - **Area**: Total drainage area at the outlet node, in km²

- **Strahler Order:** indicates the order of the highest order stream within the basin

2.1.3.4.1 Creating and Removing Basins

More than one basin can be defined within a watershed object.

To add a basin:

1. Ensure that the Channels object is displayed in a 2D view.
2. Select the outlet node for the watershed. The outlet can be any channel node.

Note: To select an outlet node that is currently defined as a watershed node (surrounded by a circle), click anywhere within that circle. To choose another channel node that is within the circle of a watershed outlet node, uncheck the **Target Outlets Visible** box in the **Display** tab of the Channels object's **Properties** dialog before selecting the desired node.

3. Select the **Add Basin** command from the shortcut menu. A new watershed boundary will appear in the active view. A new basin tab will appear in the watershed **Properties** dialog.

To remove a basin:

1. Select the **Remove** command from the Basin object's shortcut menu, or select the basin object in the WorkSpace and choose **Edit→Remove** from the menu bar.

2.2 HYDROLOGIC TOOLS

2.2.1 Watershed Tools

There are a variety of watershed tools available in the Green Kenu environment. These hydrologic tools are used in conjunction with the EnSim Core tools to provide a greater understanding of the physical characteristics of the watershed.

The Watershed Tools are:

- **Extracting Drainage Area;**
- **Extracting Drain Direction;**
- **Extracting Depression Fill;**
- **Extracting Average Upslope Elevation;**
- **Extracting Average Upslope Slope;**
- **Extracting Wetness Index;**
- **Extracting Stream Power;**
- **Extracting Relief Potential;**
- **Extracting Upstream Network;**
- **Extracting Downstream Reach;**
- **Extracting Basin Network;**
- **Extracting a Hypsographic Curve;**
- **Extracting Basin Flow Path Distances;**
- **Drainage Area Ratio Analysis;** and.
- **Slope Analysis;**

2.2.1.1 Extracting Drainage Directions

The flow direction at each node or point of the DEM can be extracted as a surface. Select the DEM in the WorkSpace and select **Tools**→**Watershed**→**Extract Drainage Direction** from the menu bar. The drainage directions can be viewed in a 2D or 3D view.

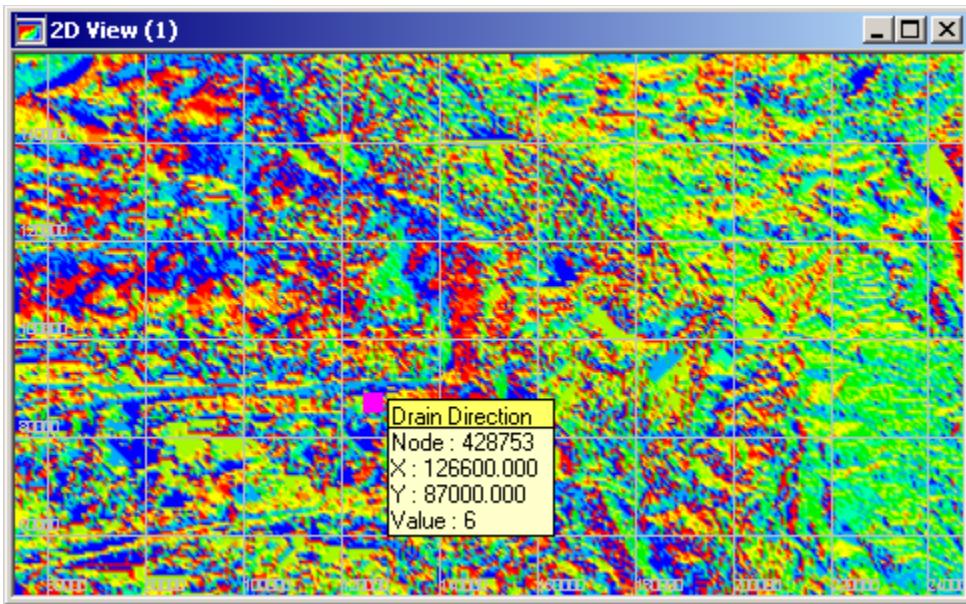


Figure 2.19: Each node in this object shows the direction in which it drains

The drainage direction values at each node correspond to the flow directions identified in the direction schematic shown below. North is assigned a value of 6.

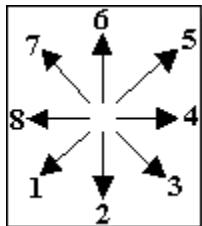


Figure 2.20: Each value corresponds to a direction, as shown above

2.2.1.2 Extracting Drainage Area

The area of upstream drainage at each node or point of the DEM can be extracted as a surface. Select the DEM or a basin in the WorkSpace and select **Tools**→**Watershed**→**Extract Drainage Area** from the menu bar. If a basin is selected, the drainage area value of nodes outside the basin boundary will default to zero. The drainage areas can be viewed in a 2D or 3D view.

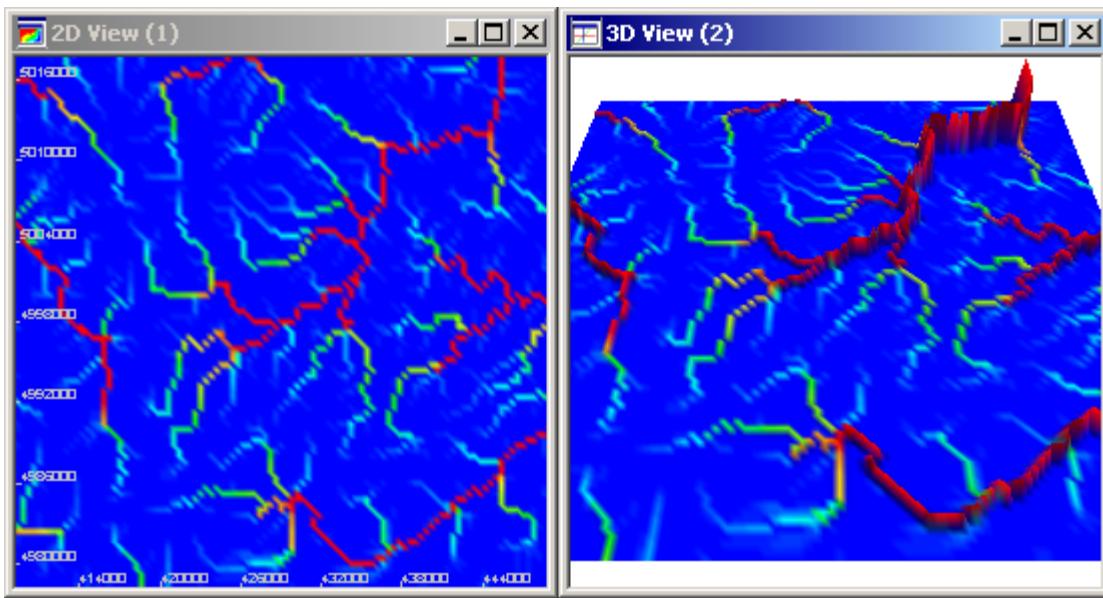


Figure 2.21: These images depict a Drainage Area map, shown in 2D (left) and 3D (right) view

2.2.1.3 Extracting Depression Fill

The modifications made to the original DEM, or grid, to create a Depressionless DEM can be viewed by *extracting the depression fill*. This operation can only be performed if the watershed was delineated using the Jenson flow algorithm. See "Creating a New Watershed Object" under Watershed Objects, on p. 136, for more information.

The depression fill shows the amount by which Green Kenue had to fill the depressions to bring them up to the level of the surrounding land. The depression fill can be extracted as a surface and displayed in a 2D or 3D view. Select the DEM or a basin in the WorkSpace and select **Tools**→**Watershed**→**Extract Depression Fill** from the menu bar. If a basin is selected, the fill value of nodes outside the basin boundary will default to zero.

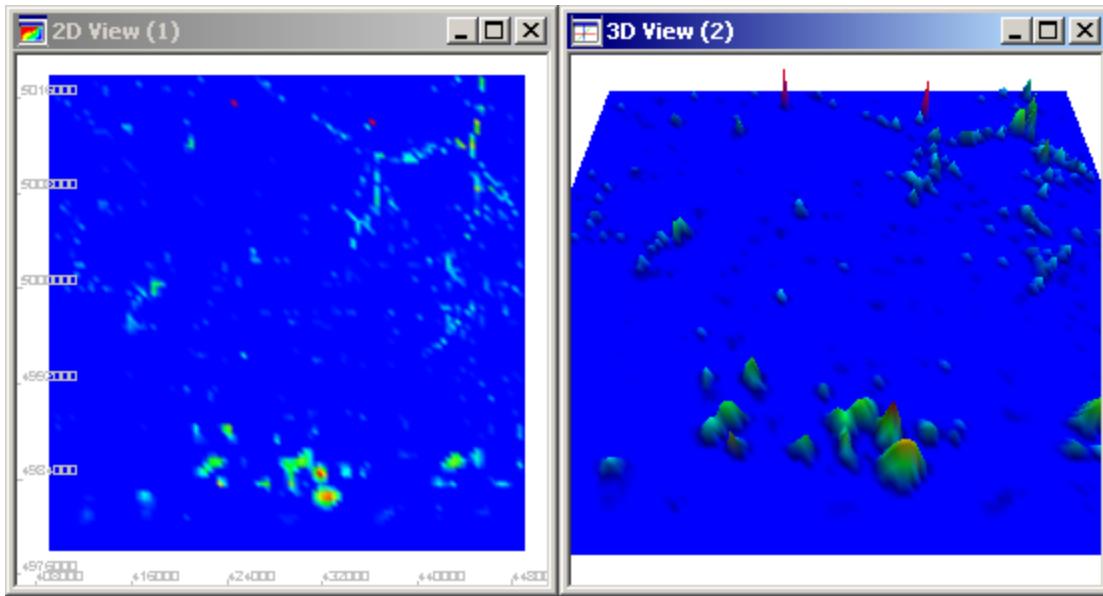


Figure 2.22: These images show a Depression Fill map in 2D (left) and 3D (right) view

2.2.1.4 Extracting Average Upslope Elevation

The average elevation of all upstream nodes flowing into each node of the Watershed DEM can be extracted as a surface. Select the DEM or a basin in the WorkSpace and select **Tools→Watershed→Extract Ave Upslope Elevation** from the menu bar. If a basin is selected, the upslope elevation value of nodes outside the basin boundary will default to zero. The average upslope elevations can be viewed in a 2D or 3D view.

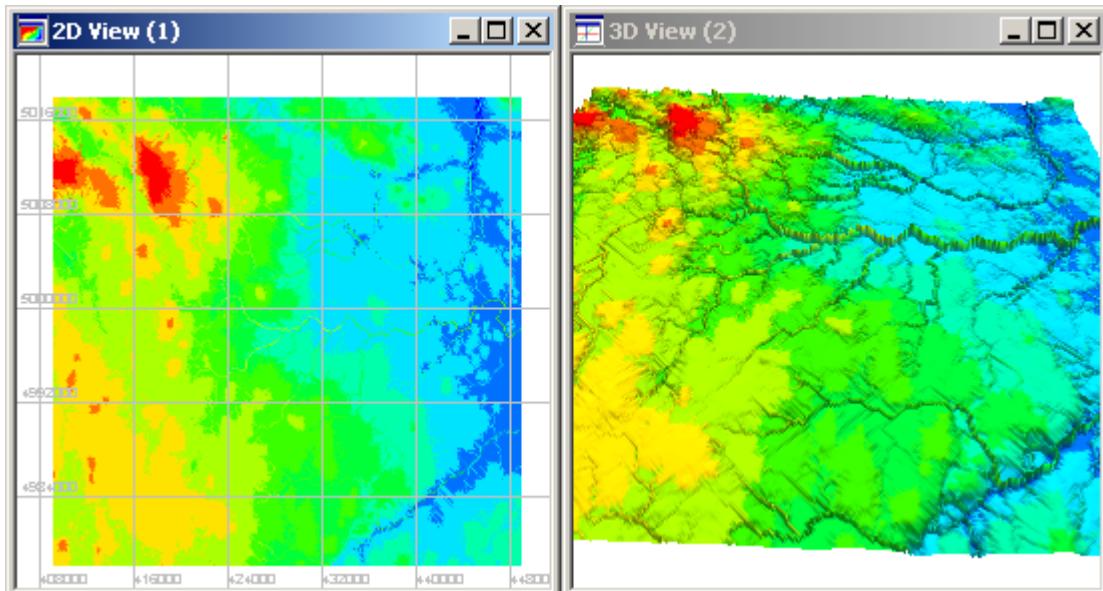


Figure 2.23: These images depict an Average Upslope Elevation map, shown in 2D (left) and 3D (right) view

2.2.1.5 Extracting Average Upslope Slope

The average slope of all upstream nodes flowing into each node of the Watershed DEM can be extracted as a surface. In this case, slopes are calculated using the 8 neighbour finite difference method. Select the DEM or a basin in the WorkSpace and select **Tools**→**Watershed**→**Extract Ave Upslope Slope** from the menu bar. If a basin is selected, the upslope slope value of nodes outside the basin boundary will default to zero. The average upslope slope can be viewed in a 2D or 3D view.

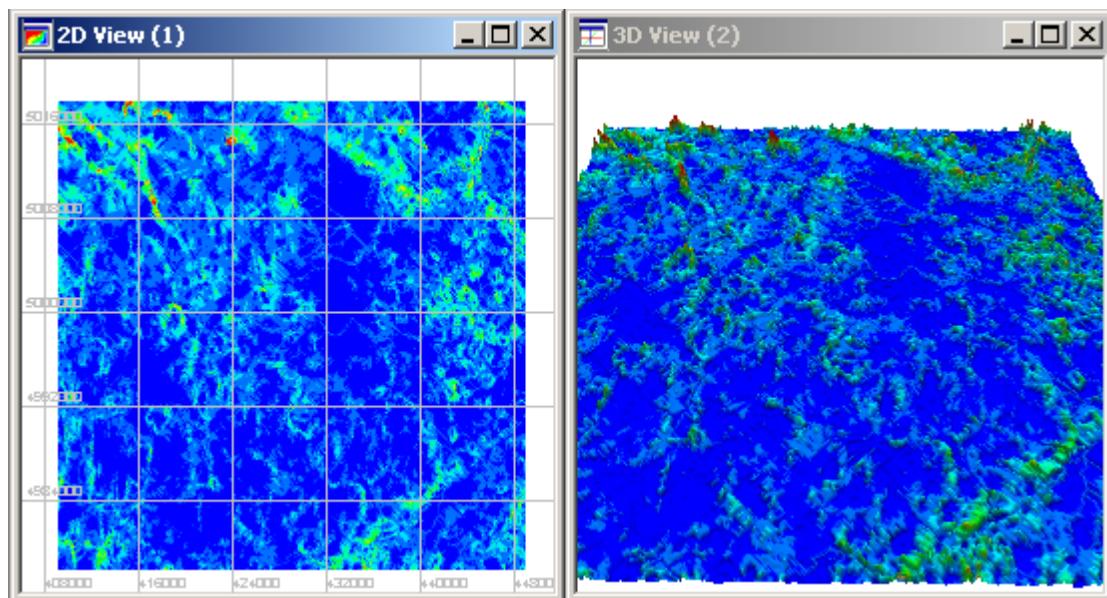


Figure 2.24: These images depict an Average Upslope slope map, shown in 2D (left) and 3D (right) view

2.2.1.6 Extracting Wetness Index

The topographic wetness index is a function of the drainage area and the slope. Higher wetness indices may indicate regions of the DEM that are more likely to generate surface runoff. These regions include both flat areas and areas with large contributing upstream drainage. The wetness index of each node of the Watershed DEM can be extracted as a surface. Select the DEM or a basin in the WorkSpace and select **Tools**→**Watershed**→**Extract Wetness Index** from the menu bar. If a basin is selected, the wetness index value of nodes outside the basin boundary will default to zero. The wetness index surface can be viewed in a 2D or 3D view.

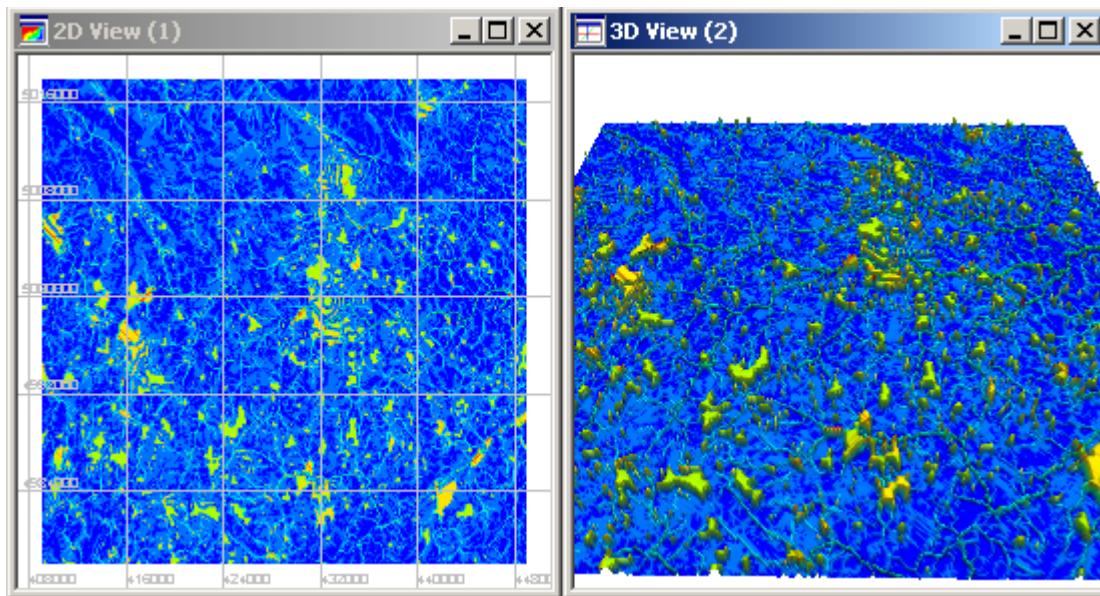


Figure 2.25: These images depict a Wetness Index map, shown in 2D (left) and 3D (right) view

2.2.1.7 Extracting Stream Power

The stream power index, like the wetness index, is a function of the drainage area and the slope. Higher stream power values may indicate regions of the DEM that are more likely to generate surface runoff. The stream power index of each node of the Watershed DEM can be extracted as a surface. Select the DEM or a basin in the WorkSpace and select

Tools→Watershed→Extract Stream Power from the menu bar. If a basin is selected, the stream power value of nodes outside the basin boundary will default to zero. The stream power surface can be viewed in a 2D or 3D view.

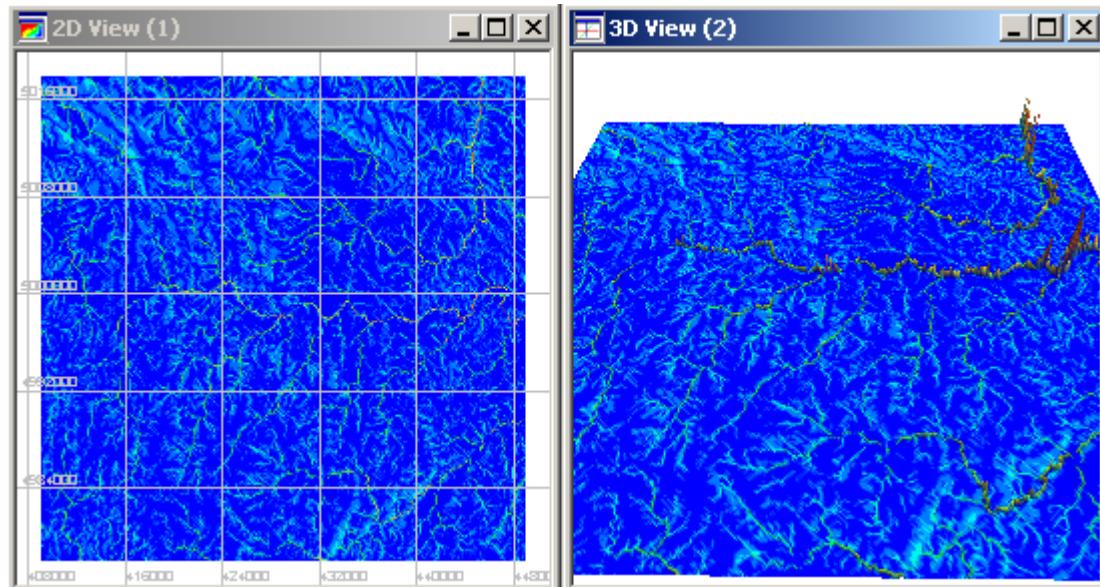


Figure 2.26: These images depict a Stream Power map, shown in 2D (left) and 3D (right) view

2.2.1.8 Extracting Relief Potential

Relief Potential is the difference between the average upslope elevation and the actual elevation. Higher values may indicate higher potential flow velocities. The relief potential of each node of the Watershed DEM can be extracted as a surface. Select the DEM or a basin in the WorkSpace and select **Tools**→**Watershed**→**Extract Relief Potential** from the menu bar. If a basin is selected, the relief potential value of nodes outside the basin boundary will default to zero. The relief potential surface can be viewed in a 2D or 3D view.

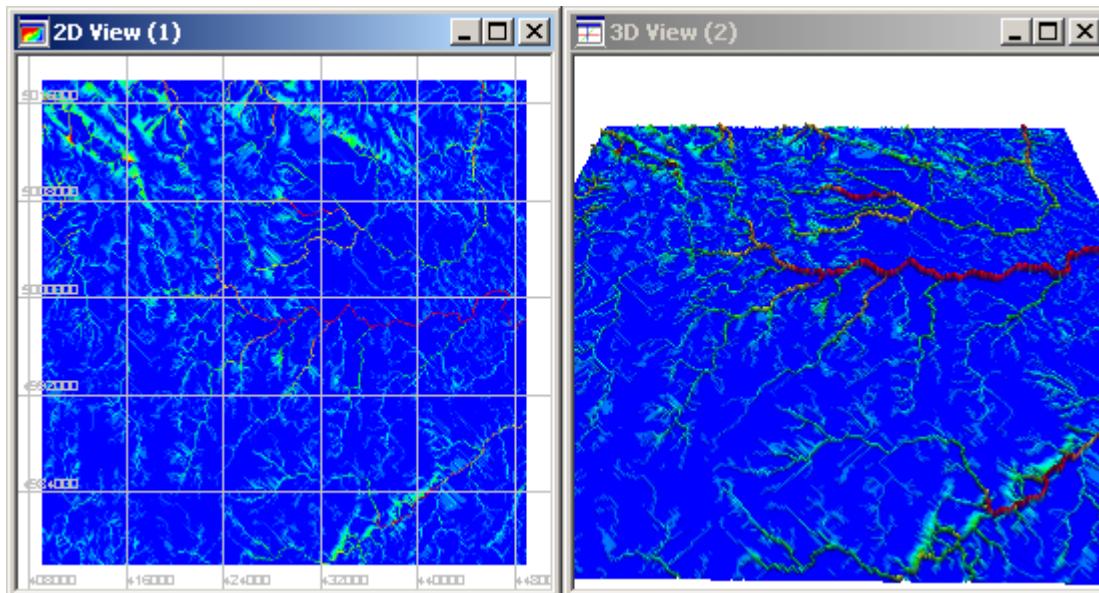


Figure 2.27: These images depict a Stream Power map, shown in 2D (left) and 3D (right) view

2.2.1.9 Extracting Upstream Network

The network upstream of a channel node can be extracted as an individual network. Select a point on a channel and then select **Tools**→**Watershed**→**Extract Upstream Network** from the menu bar.

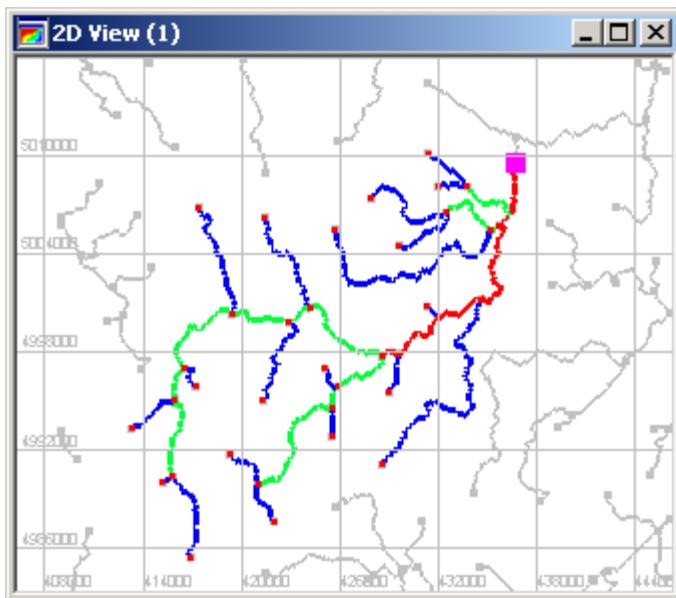


Figure 2.28: A network extracted upstream of a selected point on the channel network, shown in 2D view

2.2.1.10 Extracting Downstream Reach

The reach downstream of a channel node can be extracted as an individual network. Select a point on a channel and then select **Tools**→**Watershed**→**Extract Downstream Reach** from the menu bar.

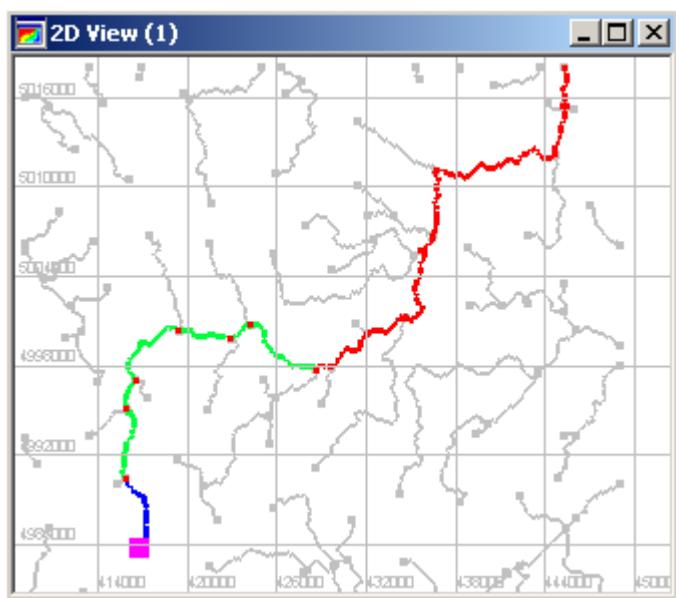


Figure 2.29: A reach extracted downstream of a selected point on the channel network, shown in 2D view

2.2.1.11 Extracting Basin Network

The channel network within a basin can be extracted as an individual network. Select the basin in the WorkSpace or a point on the basin boundary and then select **Tools**→**Watershed**→**Extract Basin Network** from the menu bar.

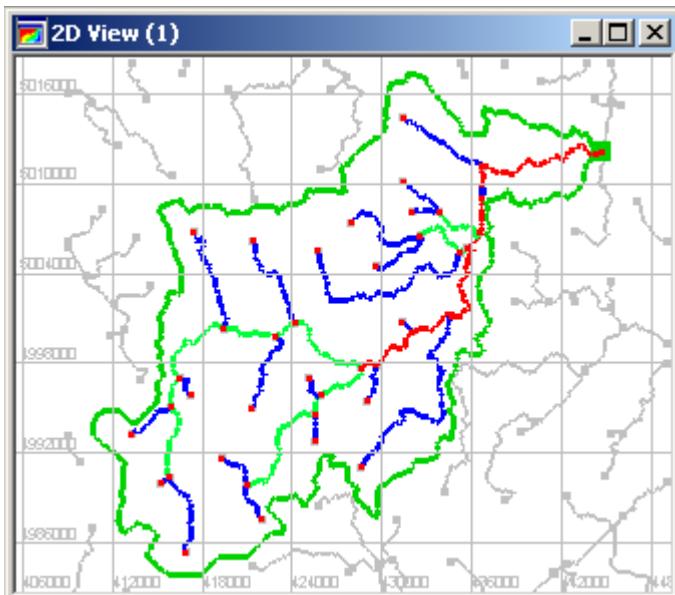


Figure 2.30: A network extracted from within a selected basin, shown in 2D view

2.2.1.12 Extracting a Hypsographic Curve

The hypsographic curve (also known as a hypsometric curve) shows the percentage of a watershed that is below an elevation. The hypsographic curve is extracted from the basin object of the watershed. Select a Basin from the Watershed in the WorkSpace and select **Tools**→**Watershed**→**Extract Hypsographic Curve** from the menu bar. The curve appears as an XY data item, which is shown as a child of the Basin object. This curve can be viewed in a 1D view.

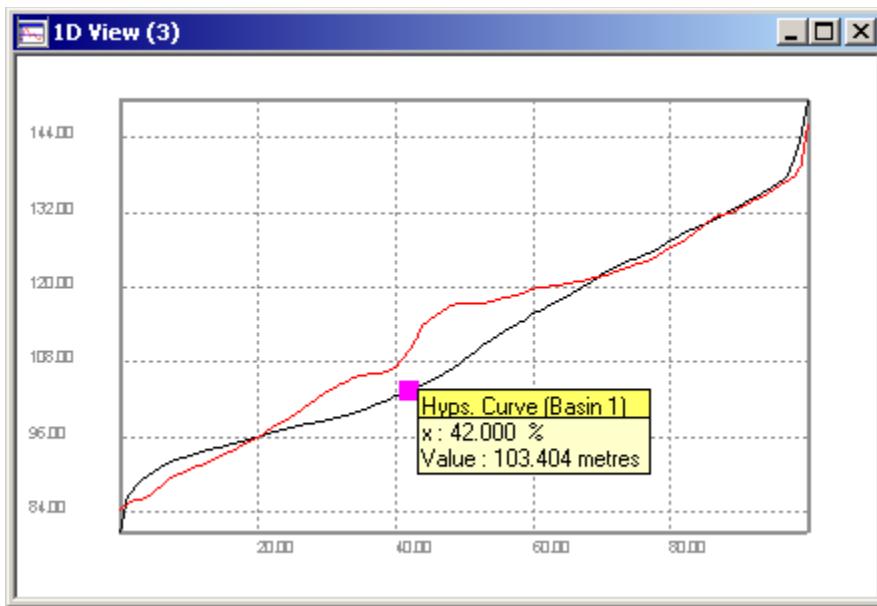


Figure 2.31: The data probe on this curve indicates that 42% of the basin is below 103.404 m in elevation

The x-axis is normalized between 0% and 100%. The curve is created by using a sorting algorithm that determines the percentage of the watershed that is below a given elevation.

2.2.1.13 Extracting Basin Flow Path Distances

The flow path distance from each nodes within a basin to the outlet of the basin can be extracted as a surface. Select the basin in the WorkSpace or a point on the basin boundary and then select **Tools→Watershed→Extract Basin Flow Path Distances** from the menu bar.

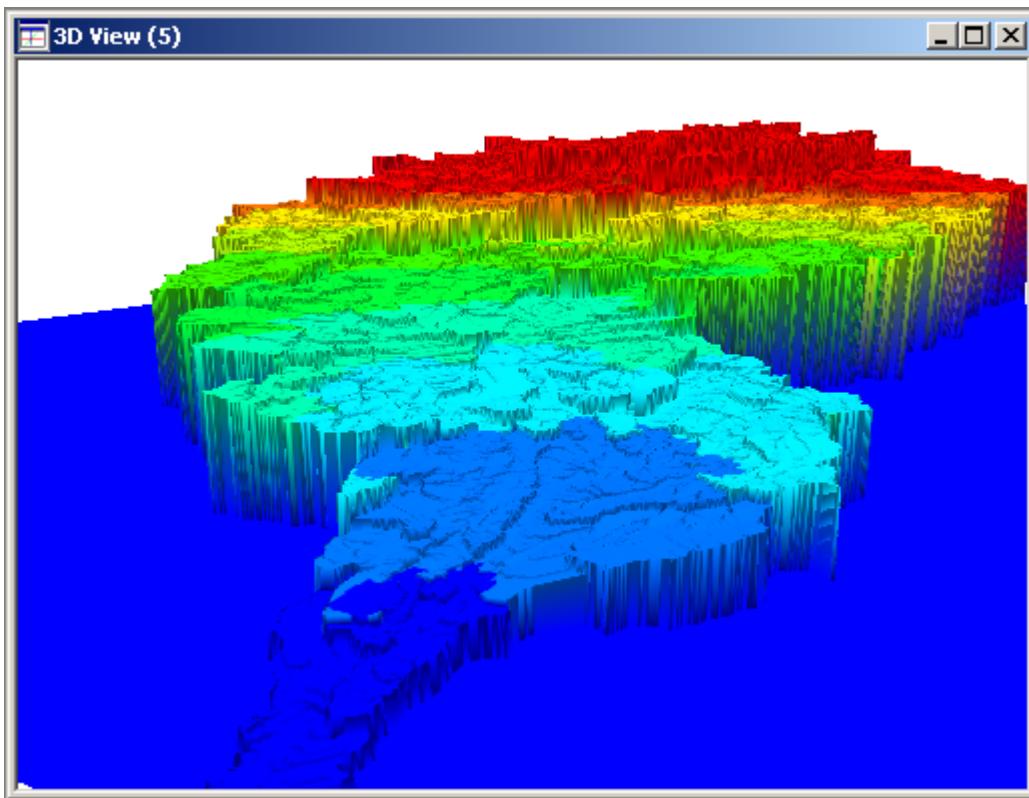


Figure 2.32: A surface representing flow path distances to the outlet extracted from a selected basin, shown in 3D view

2.2.1.14 Drainage Area Ratio Analysis

The Drainage Area Ratio (DAR) Analysis tool allows for the estimation of channel flow at a selected point, or along a user-selected section of the channel. Calculations are made using the Drainage Area Ratio method. This method assumes the flow along a channel is proportionally linear to the drainage area.

To launch the DAR Analysis:

1. Click on the watershed object in the WorkSpace.
2. Select **Tools**→**Watershed**→**DAR Analysis** from the menu bar.

This will add a DAR analysis point set object to the watershed object. The DAR Analysis will appear as an additional tab in the watershed **Properties** dialog.



Figure 2.33: The DAR Analysis tab appears after an analysis has been run

The DAR Analysis tab has two unique tabs:

- **Known Flow**

- **Computed Flow**

To identify the computed flow stations from the known flow stations, refer to the **Display** tab. Within the Display tab a unique colour may be selected for each of the stations.

2.2.1.14.1 Known Flow

The **Known Flow** tab describes known flow stations located on the channel object of the watershed.

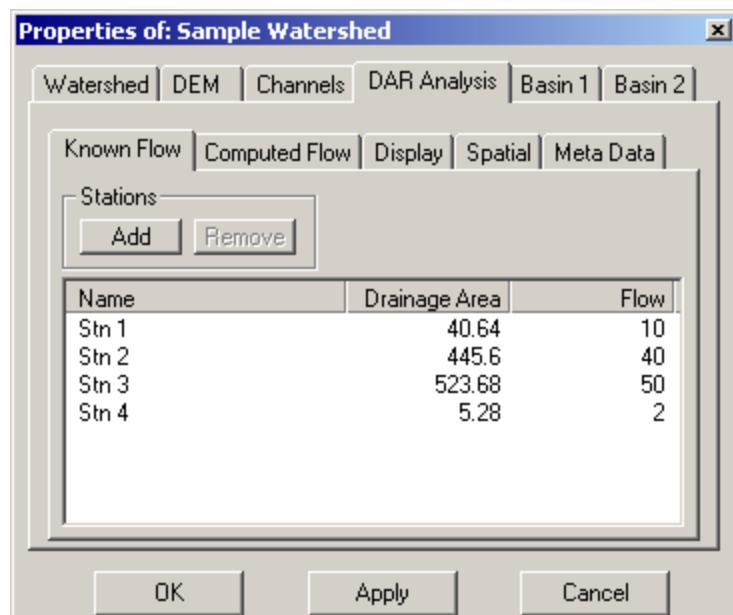


Figure 2.34: Use the Known Flow tab to enter collected data about nodes

To add a known flow station:

1. Drag the Channels object into a view.
2. With the **Properties** dialog open, click on the Channels object within the view to identify the location of a flow station.
3. Select the **Add** button from the Stations area of the **Known Flow** tab.
 - The station name will appear under the Name column. To change the name of the station, click on the name and edit the text.
 - The Drainage Area will be calculated from the station and appear in the Drainage Area column.
4. Click on the **Flow** box and enter the flow for the station.
5. Select the **Apply** button.

To remove a known flow station:

1. Click on the station on the **Known Flow** tab of the **Properties** dialog.
2. Click the **Remove** button.

2.2.1.14.2 Computed Flow

The **Computed Flow** tab describes the computed flow calculated from the known flow stations.

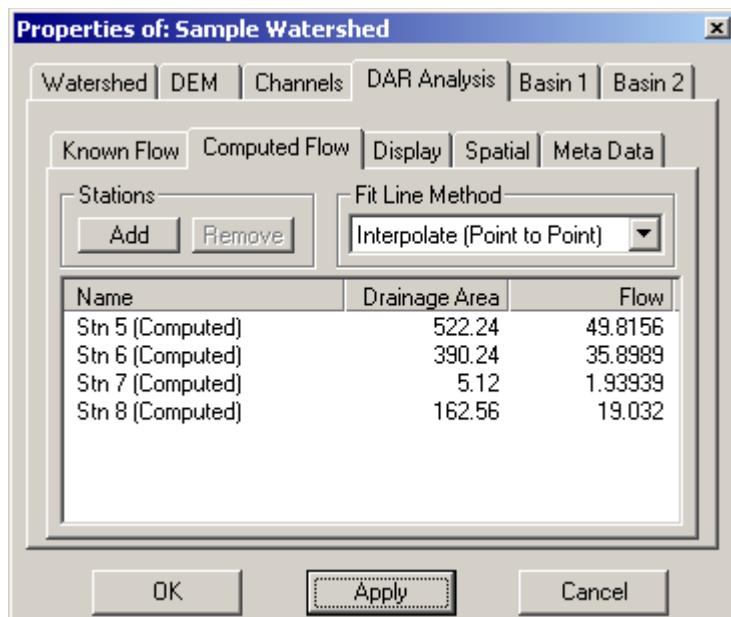


Figure 2.35: The Computed Flow is based on the known flow stations

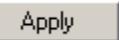
There are two methods for calculating the flow at a station along the channel:

- **Interpolate (Point to Point):** This option estimates the low flow by linearly interpolating between the nearest known lower drainage area and stream flow, and the nearest known higher drainage area and stream flow.
- **Interpolate (Average Slope):** This option estimates the stream flow along the channel by generating an average low flow versus average drainage area ratio.

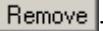
Note: Only one station is required for these calculations, as all interpolation lines are drawn through the point where drainage area and low flow are equal to zero.

To add a computed flow station:

1. Drag the Channels object into a view.
2. With the **Properties** dialog open, click on the Channels object within the view to identify the location of a flow station.
3. Select the **Add** button from the Stations area of the **Known Flow** tab.
 - The station name will appear under the Name column. To change the name of the station, click on the name and edit the text.
 - The Drainage Area will be calculated from the station and appear in the Drainage Area column.
4. The flow will be calculated from the known flow stations and appear in the flow column of the dialog.

5. Select the  button.

To remove a known flow station:

1. Click on the station on the **Computed Flow** tab of the **Properties** dialog.
2. Click .

2.2.1.15 Slope Analysis

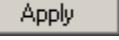
The Slope Analysis tool allows for the estimation of average slope along a user-selected section of the channel.

To launch the Slope Analysis Dialog:

1. Click on the channel object in the WorkSpace.
2. Select **Tools**→**Watershed**→**Slope Analysis** from the menu bar.

This will launch the dialog.

Ensuring the channel object is in either a 2D or 3D view,

1. In the view, click on a point that represents either the upstream or downstream boundary of a channel section of interest.
3. Select the **Point 1** button.
4. In the view, click on the other boundary point of the channel section of interest. Make sure this point is either downstream or upstream of point 1.
5. Select the **Point 2** button.
6. Select the  button. The average slope along the selected channel section will automatically be computed and displayed within the dialog.

An example is shown below:

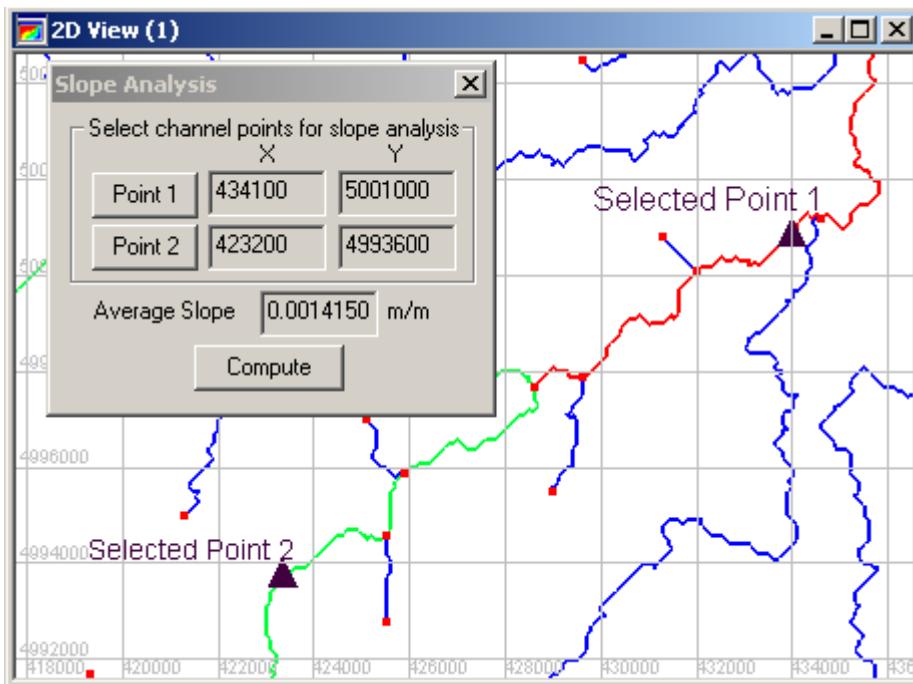


Figure 2.36: The Slope Analysis dialog and selected channel points within a view

2.2.2 Rating Curve Analysis (RCA)

2.2.2.1 Background and Theory

A rating curve is a fitted curve that approximates a discharge-or-flow versus stage-or-level relationship at a location of a particular river or stream. Green Kenue provides two curve-fitting schemes for performing a rating curve analysis: Power and Polynomial.

2.2.2.1.1 Power Curve Fit

The power curve is described by the following power equation:

$$Q = C(H - H_0)^n \quad (1)$$

where:

Q = discharge,

C = discharge when $(H - H_0) = 1.0$,

H = stage,

H_0 = stage when discharge equals 0.0, and

n = slope of the rating curve.

2.2.2.1.2 Polynomial Curve Fit

The polynomial curve is described by the following equation:

$$(Q)^{1/2} = H + C_1 \times H^1 + \dots + C_{n-1} \times H^{n-1} + C_n H^n \quad (2)$$

where:

Q = discharge,

H = stage,

C = polynomial coefficient, and

n = polynomial order.

2.2.2.2 The Rating Curve Analysis Interface

A rating curve analysis can be performed on any pair of level and discharge time series. For the sake of convenience, if a HYDAT station with both flow and level time series is selected, an RCA can be created. The HYDAT database contains discrete values (daily, weekly, monthly, and so on), and EnSim does not interpolate between data records. So, a rating curve analysis can only be performed for time periods when both flow and discharge records exist.

Note: If a time series has been restricted to a subset, the RCA will only be performed on the data included in the subset. If the **subsets** of the level and discharge time series have no dates in common, the RCA cannot be performed for those time series. It's a good idea to reset the subsets on both series before performing an RCA, and then restrict the data of the RCA afterwards. This process is explained in the section "Working With Rating Curves", on p. 169.

To perform an RCA on a HYDAT station:

1. Right-click on a HYDAT station in the WorkSpace.
2. Select **Rating Curve Analysis** from the shortcut menu. The RCA will appear in its own view window. A normal rating curve analysis cannot be performed on a HYDAT station which is missing either the level or flow time series.

If the HYDAT station does not have any level and discharge time series with overlapping dates, an RCA can be performed on any two concurrent time series, even if they're from different stations.

To create an RCA from any two time series:

1. Ensure that there is at least one Type 1 (scalar implicit; see "Native Data Items", on p. 10) time series in the WorkSpace.
2. Select **Tools→Rating Curve Analysis** from the menu bar. The Rating Curve Analysis dialog will appear.

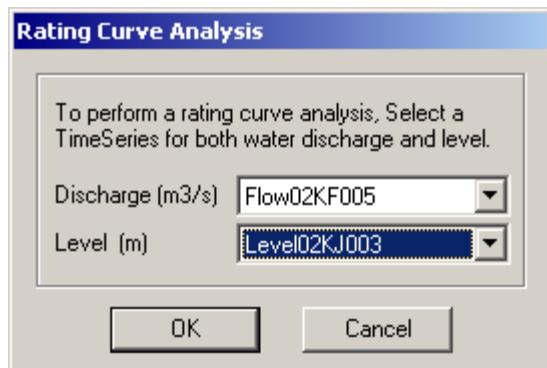


Figure 2.37: Use this dialog to select two time series for an RCA

3. Select the discharge object from the available time series in the first list box.
4. Select the level object from the available time series in the second list box.
5. Click **OK**.

Note: Although the time series in Figure Figure 2.37 are labelled as discharge and level, any two time series can be used to perform the RCA. So, any two parameters can be examined to determine their relationship. Note also that it's also possible to select the same series for both parameters.

Whatever the source of the parameters, the RCA will appear in its own window, as shown in Figure Figure 2.38.

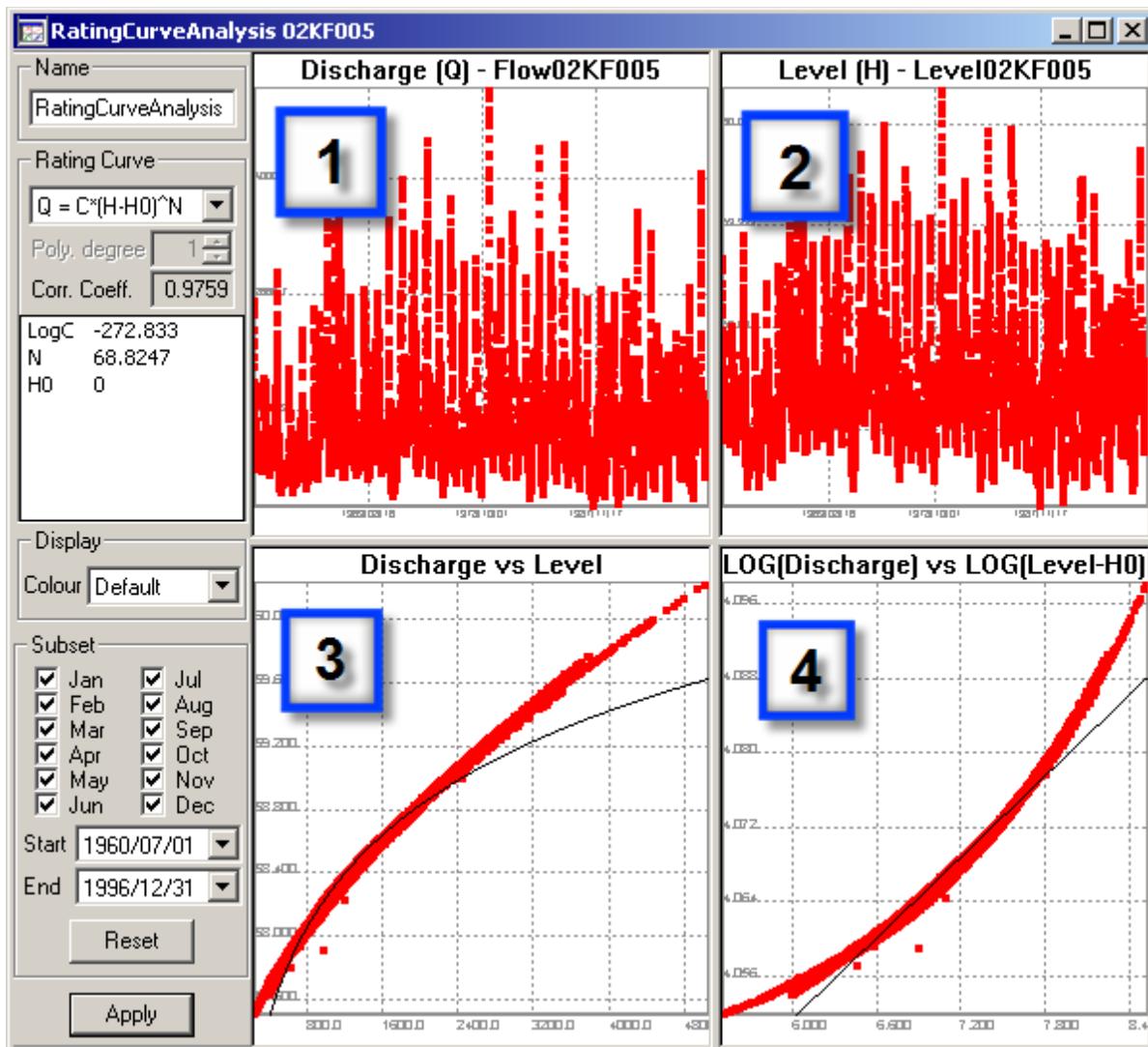


Figure 2.38: This RCA examines the flow vs. discharge relationship of the Ottawa River at Britannia from 1960 to 1996

For each RCA, four 1-dimensional views appear in the dialog:

1. Discharge (Q) time series
2. Level (H) time series
3. Discharge vs. Level
4. LOG (Discharge) vs. LOG (Level - H_0) for power curves, or SQRT (Level - H_0) for polynomial curves

The panel on the left-hand edge of the RCA window allows you to control the appearance and contents of the 1D views. Changes made within this panel are reflected in the 1D views only after you click **Apply**.

- **Name:** The name of the rating curve analysis. The default name is **RatingCurveAnalysis** followed by the Station ID.

- **Rating Curve:** The two rating curve options are listed in the top list box. Select $Q = C^*(H-H_0)^N$ for the power curve, or $\sqrt{Q} = \text{func}(H)$ for the polynomial curve. The default option is the power curve.
- **Poly. degree:** This option is only available if the polynomial rating curve has been selected. The highest polynomial order is 6. The default value is 1 to describe a line.
- **Corr. Coeff.:** This is the correlation coefficient for the fitted curve. The closer the coefficient is to 1, the better the fit of the curve to the raw data.
- The text box located below the Corr. Coeff. contains all of the coefficients for the selected rating curve. To edit any of the coefficients, click on the coefficient name. The value will become highlighted, and can then be edited. When any coefficient is changed, the rating curves will be redrawn and the correlation coefficient recalculated.
- **Display:** The data shown in the views can be examined in several colour schemes:
 - **Default:** All data points in all graphs are shown in red.
 - **By Month:** Each month of the year is shown in a different colour.
 - **By Year:** Each year is shown in a different colour.
- **Subset:** This area is similar to the subset tab from the HYDAT Properties dialog. It can be used to restrict the calculations performed in the RCA to a temporal subset of the data. Any data not included in the subset will be shown in grey.

After you have made changes to the RCA criteria, click  to adjust the data.

2.2.2.2.1 Working With Rating Curves

There are several ways to improve the fit of the rating curves. These methods include creating a subset, inactivating individual points, or directly adjusting the rating curve coefficients.

To adjust the rating curve by creating a subset:

1. Set the colour display so that the data is distinguished **By Month** or **By Year**.
2. Select an appropriate temporal subset to be examined. See "Subset" under HYDAT DATABASE, on p. 201, for more information on creating a subset of data.
3. Click . The rating curve and correlation coefficient values will be updated, and any data outside the subset will now be displayed in grey.

To deactivate an individual data point:

1. Double-click on a point to be removed from the view. The selected point will appear highlighted in magenta in all four views. A popup will appear, showing the attributes of the selected point in the selected view.
2. Right-click on the point and select the checkmark next to **Active** on the shortcut menu. The point will now be grey, and the rating curve and correlation coefficient will be recalculated.
3. Repeat steps 1 and 2 until you are satisfied with the resulting curve.

To adjust the rating curve directly:

1. Choose either $Q = C \cdot (H - H_0)^N$ or $\sqrt{Q} = \text{func}(H)$ from the rating curve options.
 - If the function has been selected, adjust the **Poly. Degree** option as well.
2. Click on any of the equation coefficients listed in the text box in the left-hand panel.
3. Adjust the coefficients until an appropriate curve is determined.
4. Click **Apply**. The rating curve and correlation coefficient will be updated.

The following examples illustrate how an improved rating curve can be obtained.

Example 1

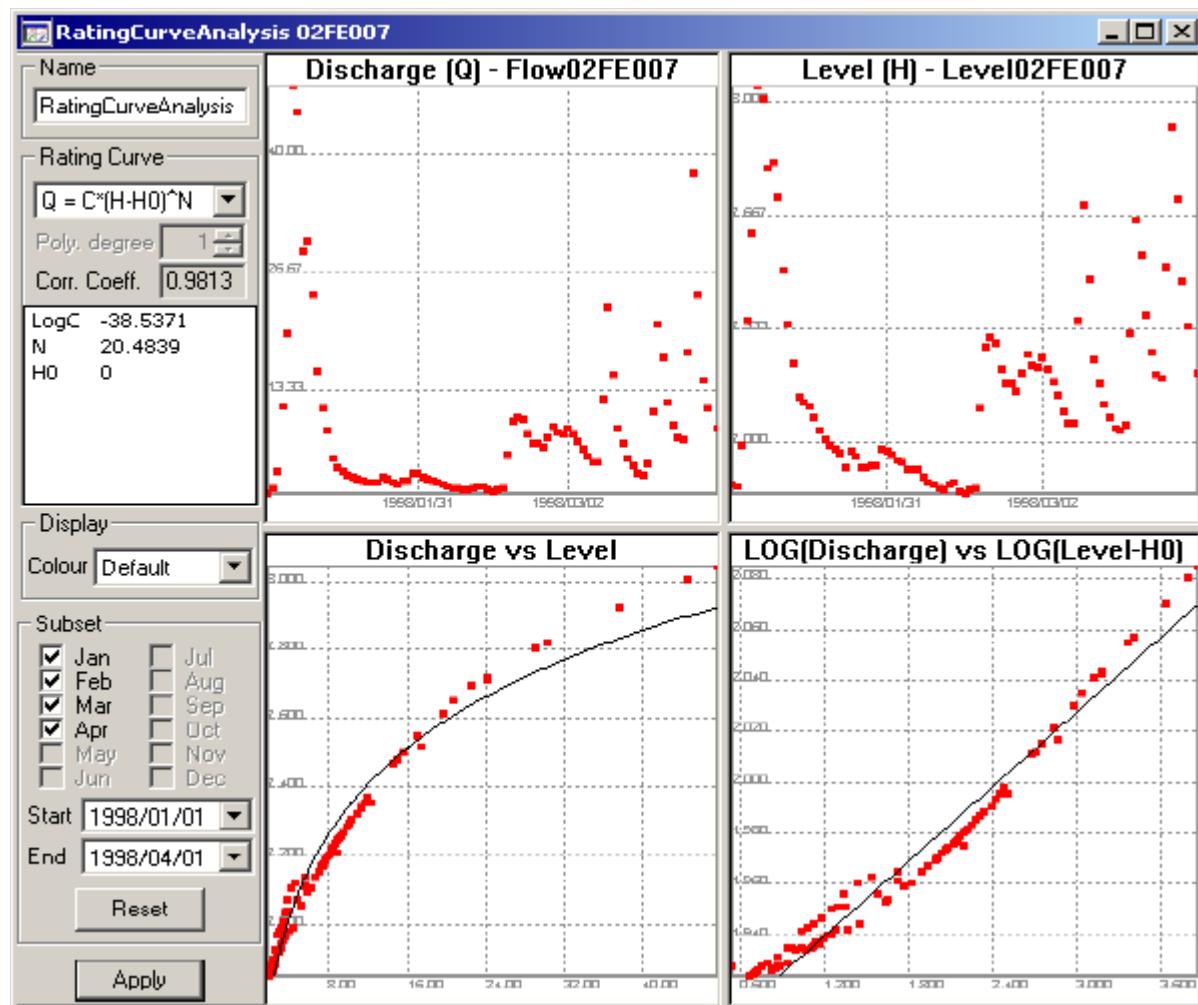


Figure 2.39: This RCA has not yet been adjusted.

The example in Figure 2.39 contains four months of data. The power rating curve is used. As shown, the correlation coefficient is 0.9813. This data set has a high density of low flow data. To remove data used in the rating curve, a subset of the data can be completed to remove a majority of the low flow data. A majority of the low flow data occurred in the months of January and February. Remove the data from the months by clicking on the check boxes

adjacent to the months in the subset box. Data points are not removed from the view, but will appear greyed out. In this example, January and February have been removed, as shown in Figure Figure 2.40.

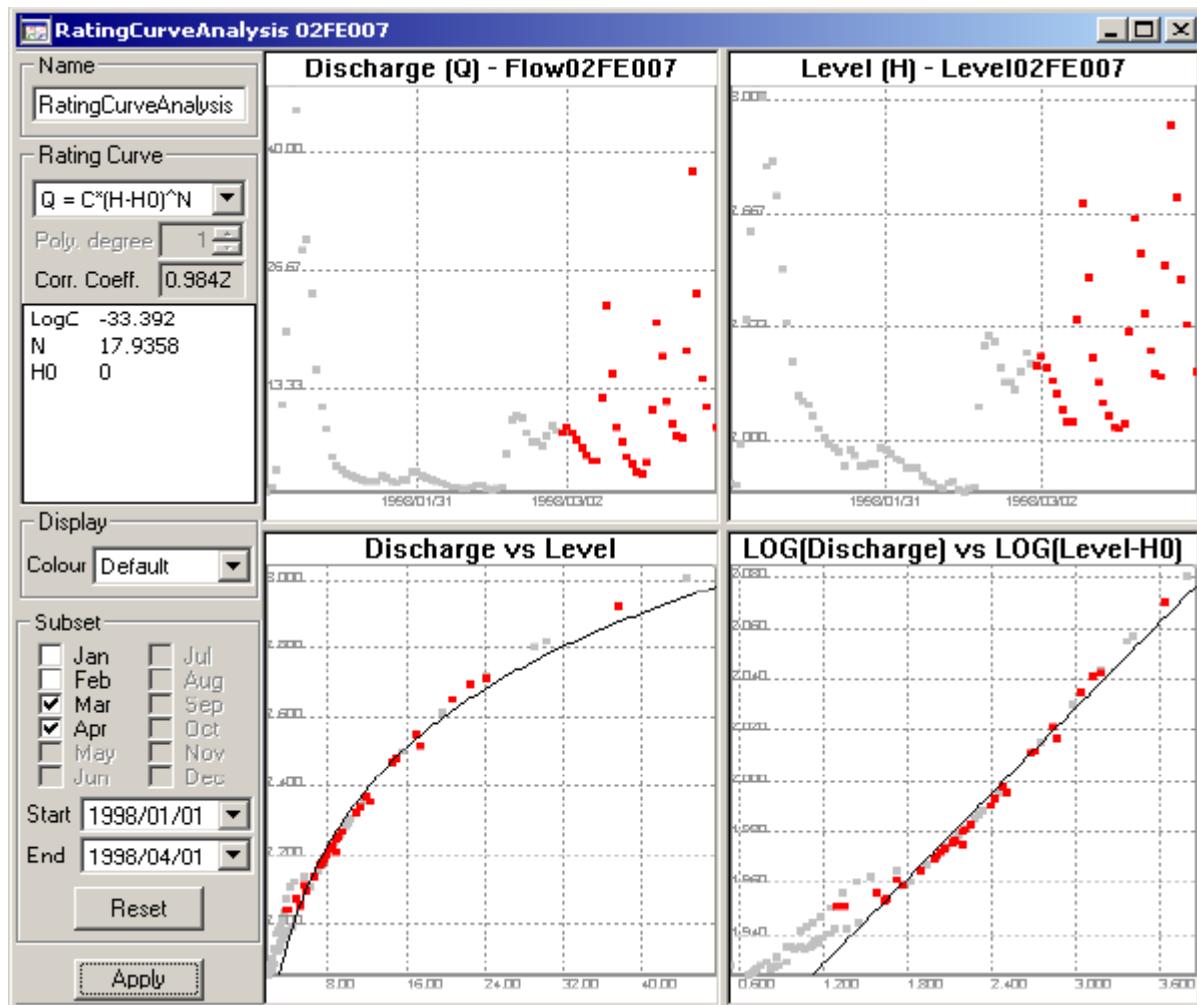


Figure 2.40: The data from the months of January and February have been removed from the calculations

By removing the selected months, a new rating curve has been calculated and the correlation coefficient has improved to 0.9842.

As shown in Figure Figure 2.40, several low flow points are skewing the upper section of the curve. An additional temporal subset can be created, but the low flow points may be in the middle of the data set. In that case, creating a temporal subset would not solve the problem. The remaining low flow data points can be removed by inactivating the points. Removing the few remaining points by selecting them and making them inactive results in a rating curve with a correlation coefficient of 0.9966, as shown in Figure Figure 2.41.

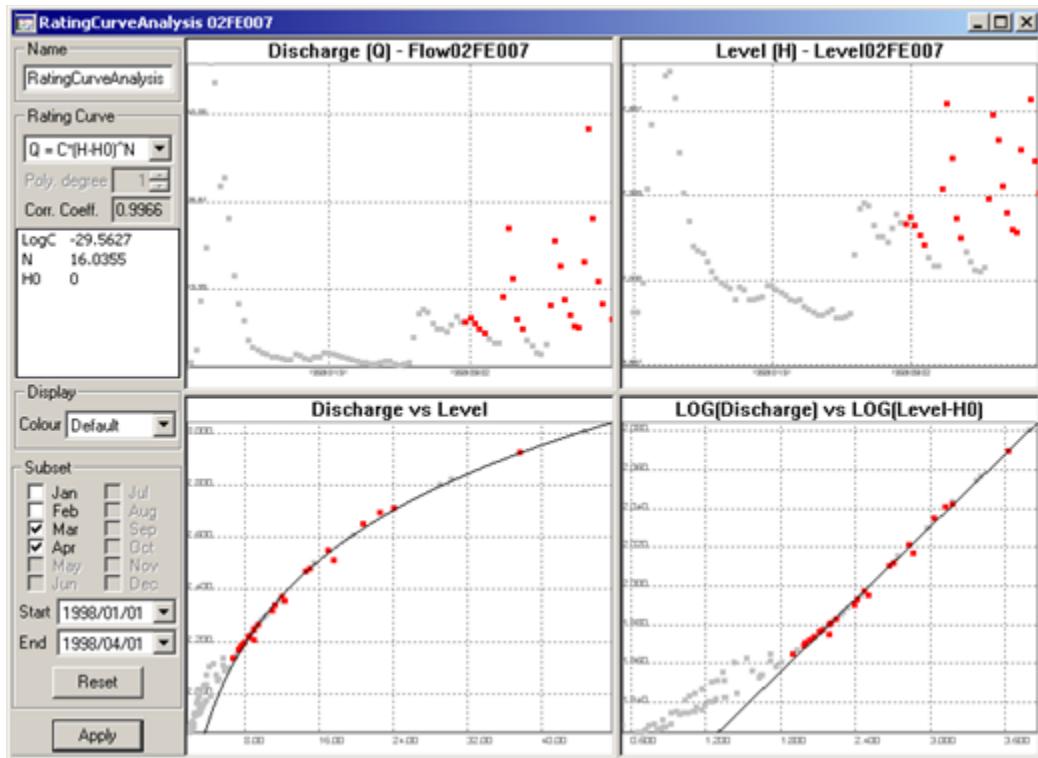


Figure 2.41: This RCA has had several low flow values inactivated, producing a well-fitting curve.

Example 2: With complex examples, it may be harder to find an appropriate rating curve.

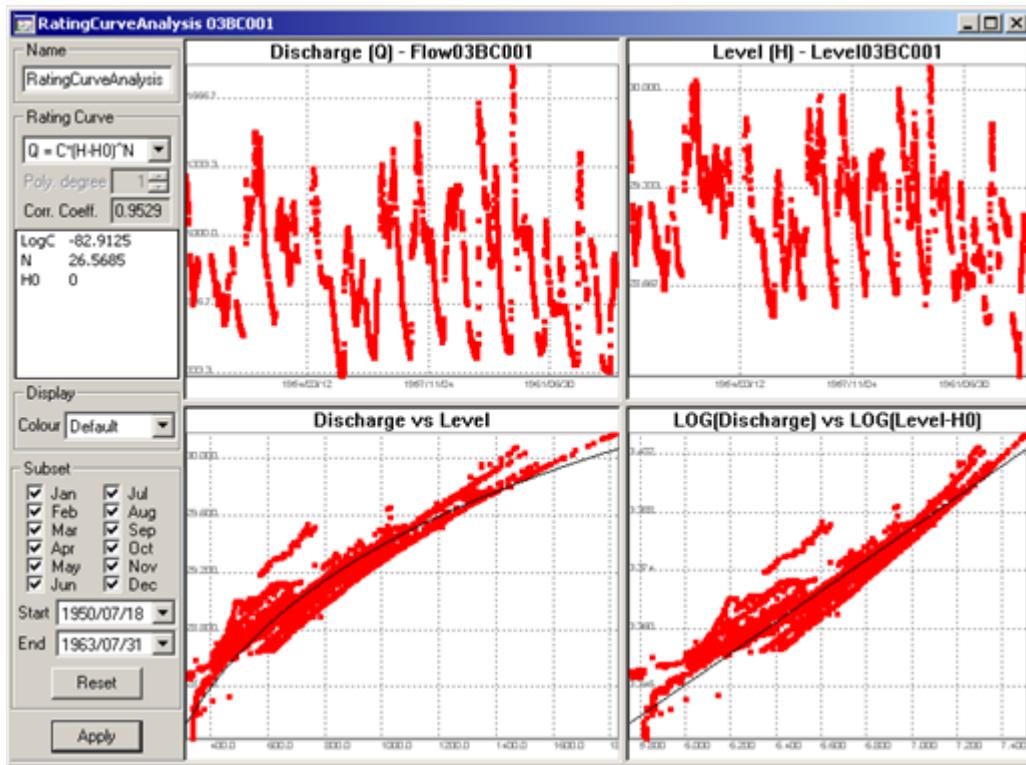


Figure 2.42: This RCA is far more complex than the previous example, and has a correlation coefficient of 0.9529

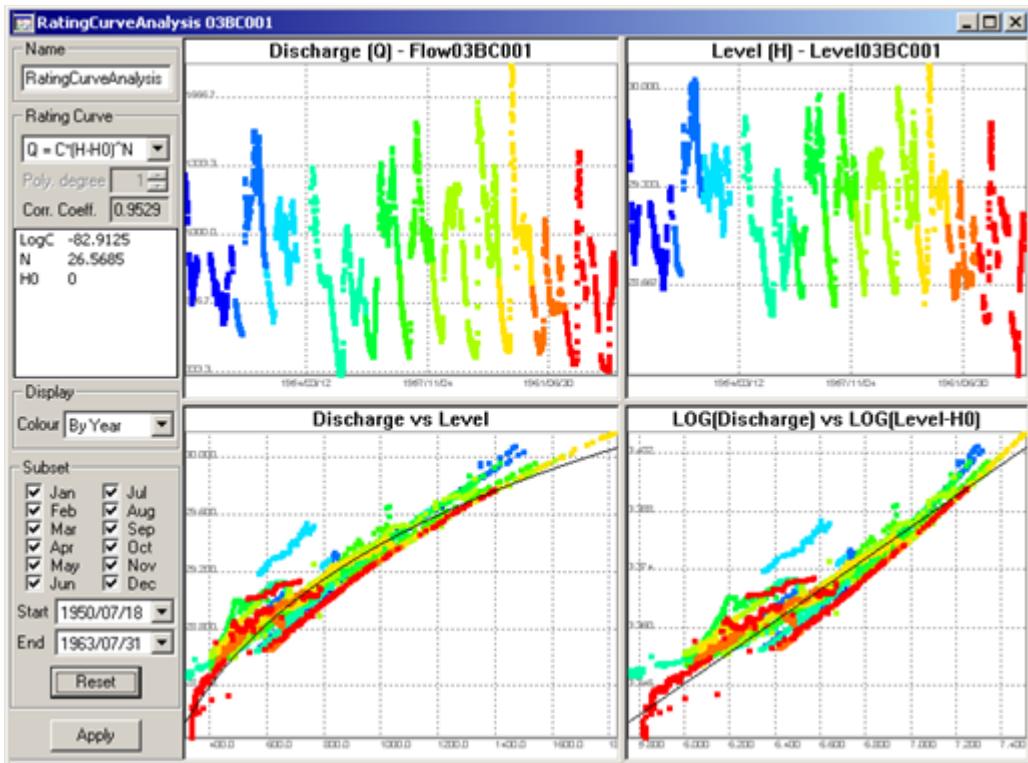


Figure 2.43: Examining the curve by year makes it clear that more than one rating curve exists

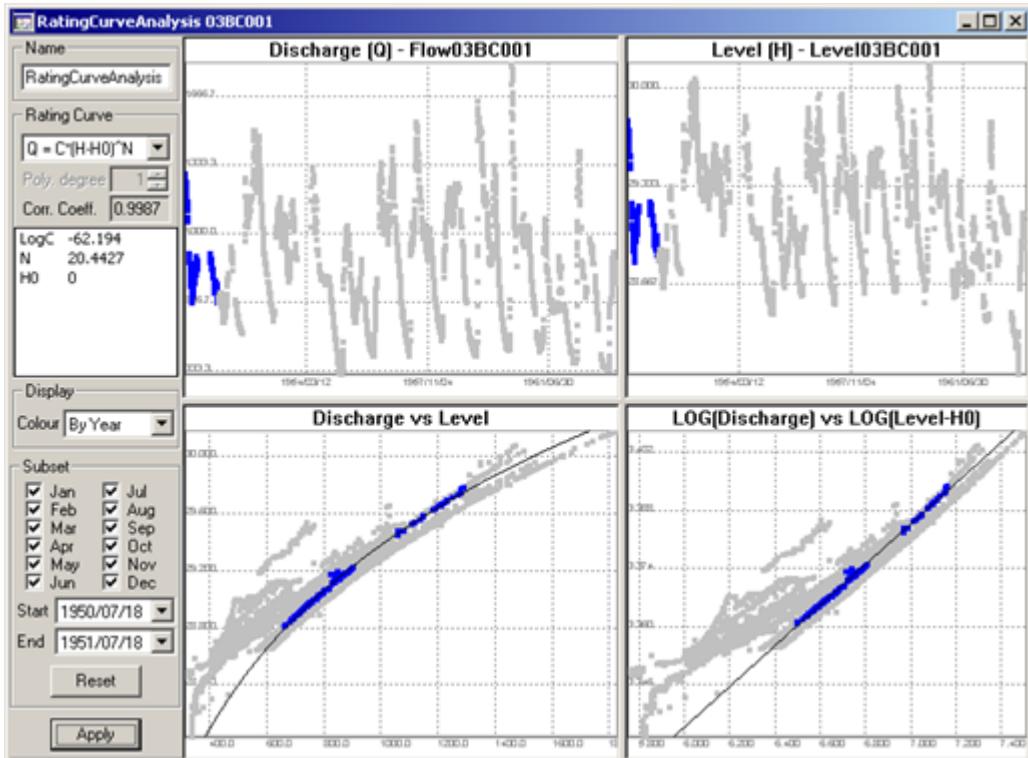
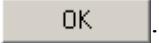


Figure 2.44: Restricting the temporal subset to a single year results in a correlation coefficient of 0.9987

2.2.2.3 **Opening an Existing RCA**

To open an RCA:

1. Select **File→Open** from the menu bar.
2. Locate and select the RCA in the dialog box.
3. Click .

2.2.2.4 **Saving an RCA**

To save an RCA:

1. Select the RCA in the WorkSpace.
2. Select **File→Save** from the menu bar. The file will be saved as an *.rca file.

2.3 WATFLOOD

2.3.1 WATFLOOD Map Files

The Watflood Map file is an input data file required by the WATFLOOD hydrologic model. The file consists of a regular grid of cells with data values for several physiographic attributes assigned to each cell.

The Watflood Map object uses information in the watershed object to calculate most of the data attributes for each grid cell. The land use data attributes are calculated using other data information and tools.

The Watflood Map will be stored in *.map format. See "Supported Foreign File Types [Green Kenuel]", on p. 311, for more information about this type of file. It can be viewed by dragging the map object into a view.

For WATFLOOD to use the map file, it must contain—at minimum—the information provided by the watershed object, in addition to land use information.

2.3.1.1 Opening an Existing Watflood Map File

To open an existing map file, select **File**→**Open** from the menu bar, or the  button on the tool bar. Map files have the file extension *.map. The watershed map file will be listed in the Workspace under the **Data Items** category and have this icon: .

Note: A basin (*.shd) file, a WATFLOOD file that represents a processed map, can also be opened and displayed in Green Kenuel, but it **cannot** be edited or saved. All changes must be made directly to the map file, and a new basin file must be generated by WATFLOOD. Basin files have this icon: .

2.3.1.2 Creating a New Watflood Map File

To create a new map file, select **File**→**New**→**Watflood Map...** from the menu bar. A Watflood Map object will be created and listed under the **Data Items** category.

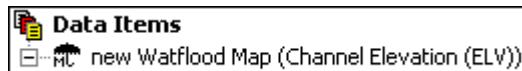


Figure 2.45: A new Watflood Map object

The property dialog box will appear. The **Map Gen** tab will appear with all the values in the **Specification** area set to zero.

The new map file does not need to be associated with a watershed. The specifications can be set manually or automatically.

To set map file specifications manually:

1. Enter **Origin**, **Count**, and **Delta** values for the cells of the grid.

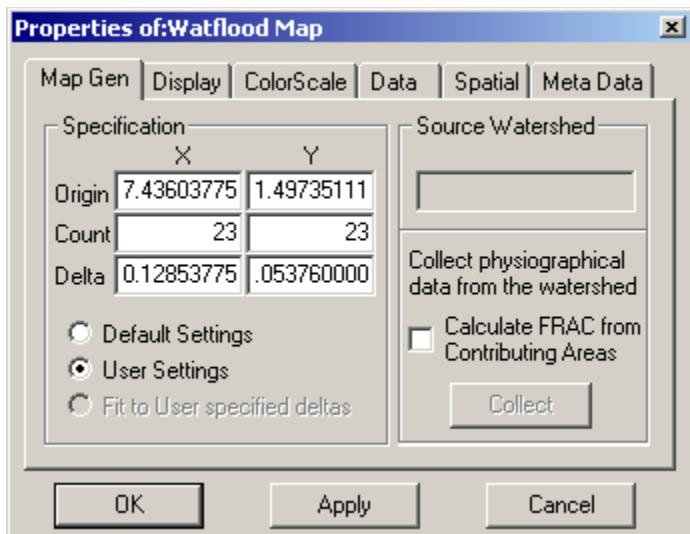


Figure 2.46: This Map Gen tab has had data entered manually

2. Click the **Apply** button.

To set map file specifications automatically:

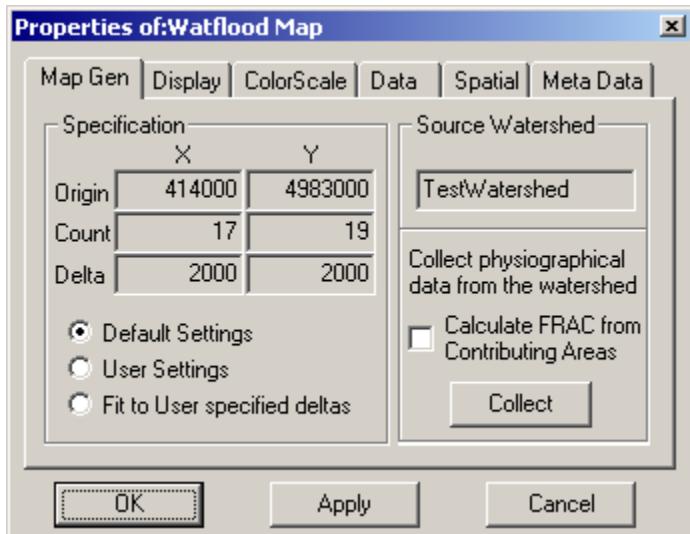


Figure 2.47: This Map Gen tab has been given values from a source watershed

1. Setting the specifications manually requires that a watershed be associated with the map file. Select the **Default Settings** option. The specifications will be determined by fitting the basin to the associated watershed. The map file will be created from all the watershed data contained within the Watflood Map specifications.

Selecting **Collect** will cause all of the watershed data contained within the WATFLOOD specifications to be extracted and applied to the Watflood Map object.

If the "Calculate FRAC from Contributing Areas" option is enabled, the effective area of each cell is adjusted based on the amount of inflow from neighbouring cells. Please see the WATFLOOD manual for further details. (Section - Grid Drainage Area (FRAC))

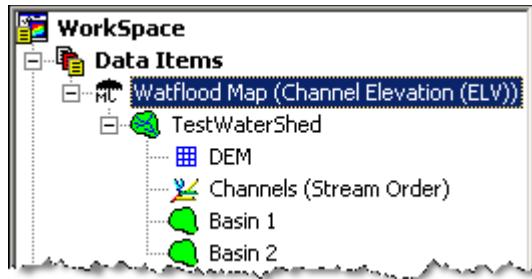


Figure 2.48: A Watflood Map associated with a watershed in the WorkSpace

Note: A Watflood Map **cannot** be created from a watershed object that has a rotated grid. Open the **Properties** dialog of the watershed object's DEM and ensure that the **Angle** field of the **Spatial** tab has a value of zero.

When you drag a watershed object into the Watflood Map, a default grid is automatically created that conforms to the following WATFLOOD grid rules:

- Each watershed outlet must be a square outside the watershed.
- There should be a border of blank grid squares around all sides of the watershed boundary. The watershed outlet may be within this border.
- The maximum grid size is 99 cells by 99 cells.

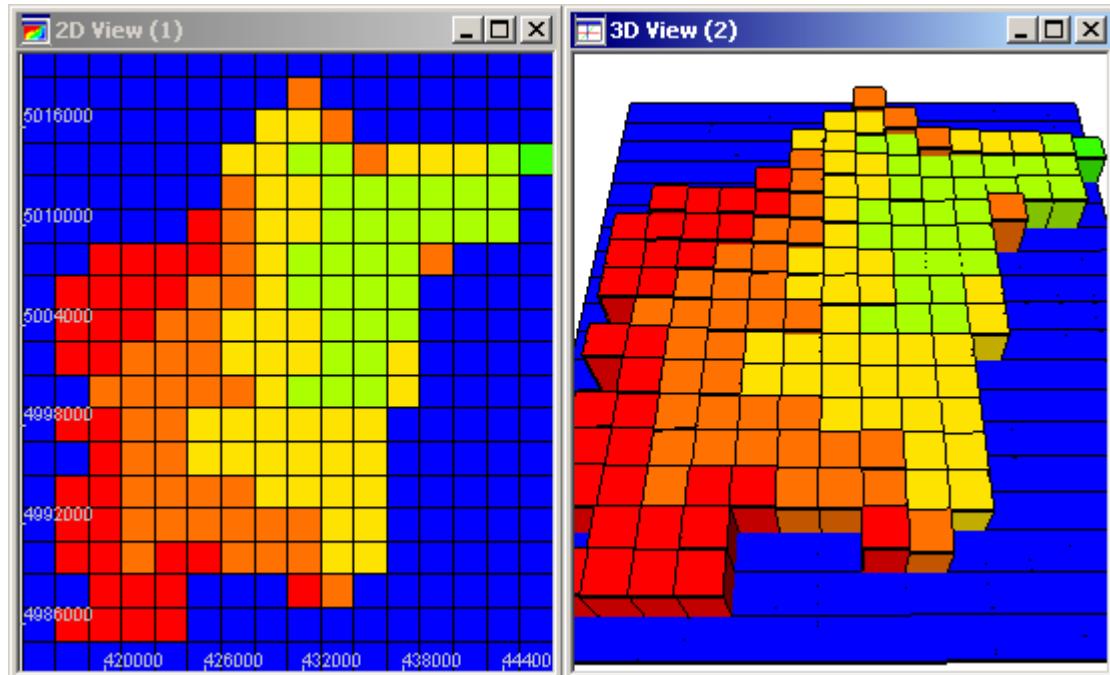


Figure 2.49: A Watflood Map grid can be displayed in either a 2D (left) or 3D (right) view

To return to the default grid:

- In the Watflood Map's shortcut menu, select **Set Spatial Default From Watershed**.

Note: All changes made to data attributes will be lost, including land class data.

2.3.1.3 Modelling Multiple Watersheds

Creating a Watflood Map for multiple watersheds that are independent of each other (i.e. their boundaries do not overlap) is straightforward. The Watflood Map will have two or more separate watersheds, based on the Basins defined in the Watershed Object.

In the following image, each separate watershed, or basin, is outlined in black. Black dots indicate the locations of each watershed outlet. Also, note that the map cells just downstream of each outlet are of the same colour, showing that each watershed outlet has a similar elevation.

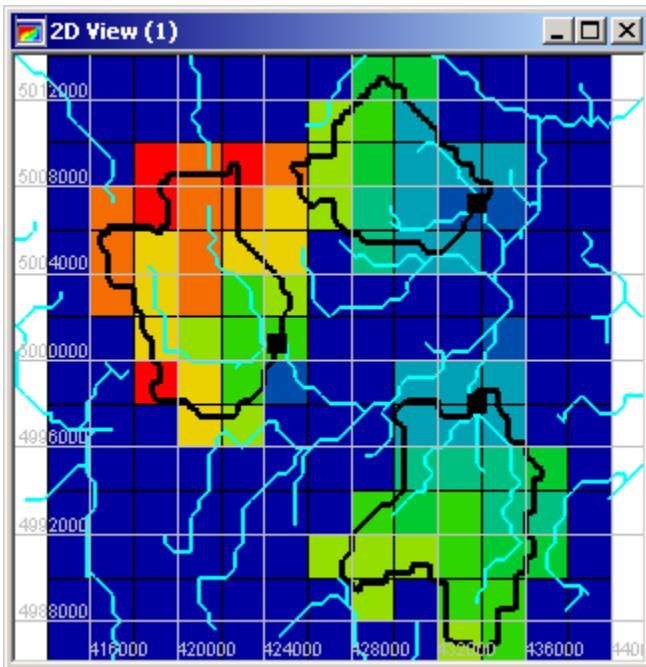


Figure 2.50: Because they do not overlap or touch, these watersheds are independent

If the watersheds being modelled are nested, with smaller watersheds lying within larger watersheds, then they are not independent, and will appear in the Watflood Map as a single watershed, as shown in Figure 2.51, below.

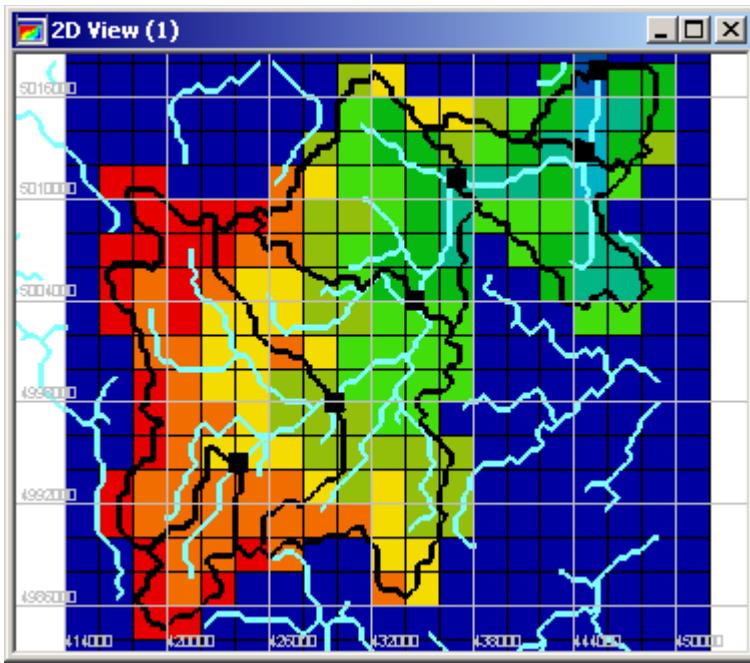


Figure 2.51: Because these watersheds are contiguous, they are treated as a single watershed

This is not a problem, since WATFLOOD models the outflow, runoff and other parameters for **each cell** within the grid. Parameters should be edited for cells that are crossed by a watershed boundary shared between two basins. See the Drainage Area parameter under "Description of Data Attributes", on p. 179, for an example of this.

2.3.1.4 Watflood Map Data Attributes

WATFLOOD data attributes are listed in the Data tab of the Watflood Map's Properties dialog. Once the Watflood Map file is associated with a watershed object (see "Creating a New Watflood Map File", on p. 175), Green Kenue automatically calculates most of the data attributes based on the watershed object. Land use data is not calculated automatically and must be added manually. The default data attributes for the Watflood Map can be edited.

2.3.1.4.1 Description of Data Attributes

The physiographic data required by the WATFLOOD hydrological model are as follows:

- **Channel elevation (ELV):** This is the elevation at the midpoint of the main channel within a cell. Note that for multiple basins with different watershed outlets, the elevation at the outlet cells must be equal. Green Kenue forces all outlet cell elevations to be equal to the lowest outlet cell elevation.
- **Drainage area (FRAC):** This is the percentage of the area of a cell within the watershed boundary that flows in the indicated drainage direction (see Drainage Direction, below). Green Kenue will assign a drainage area of 100% to all cells within the basin outline. Essentially, the entire land area of the cell is considered to flow in the stated drainage direction. Cells that lie on the basin boundary will be assigned a percentage based on the area of the cell that is included within the basin boundary.

The drainage area of a cell can be greater than or less than 100%. This accounts for cells that have multiple channels that drain into different cells. Adjustment of the drainage area of cells must be done manually.

For example, in the figure below, two channels flow through the dark-green cell. One drains to the east and the other to the south. In terms of element proportions, 35% of the cell area drains into the channel flowing south, and 65% of the cell area drains into the channel flowing east. The drainage direction is assigned as east, so the 35% of the green cell that drains south is added to the red cell, which is south of the green cell. This results in the drainage area of the green cell being 65%, and the red cell 135%. See "Editing Watflood Map Data Attributes", on p. 183, for information on changing the drainage area of a cell.

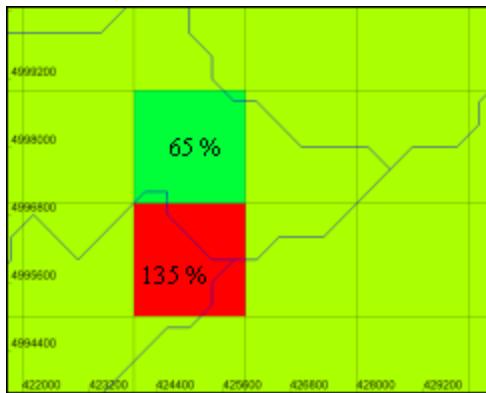


Figure 2.52: The sum of the two cells' drainage proportion is 200%

This technique can also be applied in the case of modelling multiple watersheds when a cell is split between two basins.

- **Drainage direction (S):** This value indicates the direction of the majority of flow out of the cell. Possible directions include North, North East, East, South East, South, South West, West, North West or N/A (not applicable). "Not applicable" is only applied to the cell containing the watershed outlet node. By selecting the drainage direction as the current attribute, the direction will be represented by an arrow in both the 2D and the 3D views. The direction can also be viewed when drainage direction is not the current attribute by checking the **Directions Visible** check box on the **Display** tab of the Watflood Map's **Properties** dialog

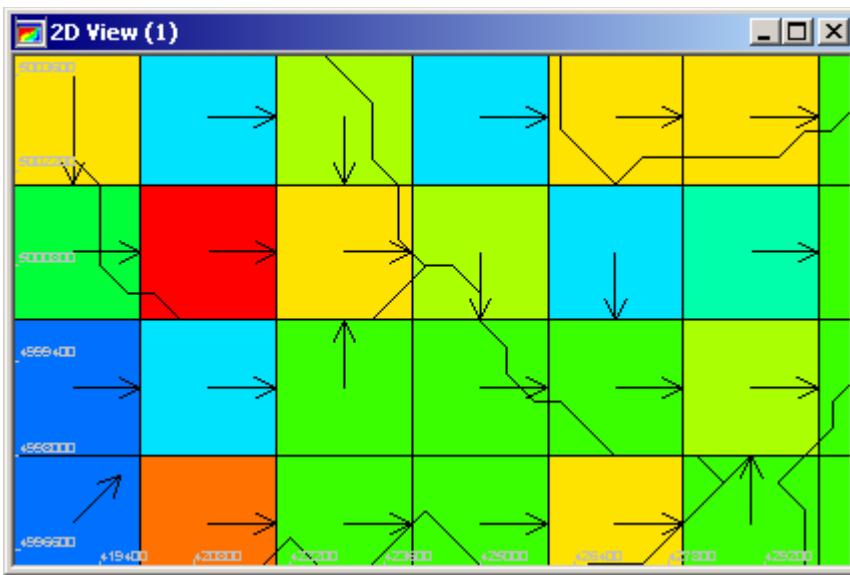


Figure 2.53: Each cell is assigned a single drainage direction value

- **River class (IBN):** This attribute defines the river roughness classes, with each class having a different value for the three roughness parameters, roughness of floodplain, channel and overland flow, defined in the WATFLOOD parameter file. Green Kenu provides a default value of 1. A maximum of five river classes can be defined.
- **Contour density (IROUGH):** This gives an indication of the number of contours in a cell. It is a relative description of the roughness within a cell. The contour elevation interval is defined at the top of the **Data** tab. The default interval is 1. The value of contour crossings in a cell has a minimum of 1 and a maximum of 99. WATFLOOD cannot handle contour densities greater than 99, so all cells having a contour density equal to or greater than 99 will be assigned a value of 99. If multiple cells have a contour crossing value of 99, increase the contour elevation interval to ensure the roughness is appropriately described.
- **Channel density (ICHNL):** This is the number of main channels that cross a grid square. The value must be between 1 and 5. If it is greater than 1, each channel is considered equal in size.
- **Routing reach number (IREACH):** Values greater than zero will output channel inflows at those cells. These inflows can be used for external routing, which may be desirable in modelling tidal or backwater effects. The default value assigned by Green Kenu to all cells is zero.
- **Land use:** The number of **Land Cover Classes** can be modified within the **Data** tab. The land use data attributes may be added manually (see "Adding Land Use Data Using Closed Polygons", on p. 183 or automatically from preprocessed GeoTIFF files (see "Adding Land Use Data Using GeoTIFFs", on p. 186.
- **Bankfull capacities:** Bankfull capacity is the maximum flow in cubic metres per second the stream in a cell can handle before flooding. It is used for animation of flooding. The **Include Bankfull Capacities** checkbox adds a blank section to the map file. Values need to be manually inputted for each cell. If the Bankfull Capacities section is not included,

WATFLOOD calculates default capacities when the watershed file is generated. If the **Include Bankfull Capacities** box is checked, but the values are not changed from the default values of zero, they are disregarded and WATFLOOD generates them automatically.

Note: The outlet cell should exist in the cell immediately downstream of the last watershed cell. The outlet cell should only have a value for channel invert elevation. All other parameters for this cell should have a value of zero.

2.3.1.4.2 Calculating the Default Data Attributes from the Watershed Object

Most of the data attributes of the Watflood Map are automatically calculated from the DEM. Exceptions include routing reach number, land use and bankfull capacities. Once the default data attributes are calculated, they should be checked for accuracy. The default values are automatically calculated for a newly created Watflood Map when the watershed object is dropped into the Watflood Map object.

To reset the data attributes to their default values, make sure that the appropriate watershed object is associated with (i.e. is a child of) the Watflood Map. Select the command **Collect Data From Watershed** from the shortcut menu of the Watflood Map.

Note: Changes to the data attributes will be lost, except for land use class attributes.

2.3.1.4.3 Displaying Different Data Attributes in the Watflood Map

To display the same data attributes for all cells:

1. Ensure that the Watflood Map is displayed in a 2D or 3D view. 2D is preferable.
2. Double-click on the Watflood Map in the WorkSpace. A Properties dialog will appear.
3. In the **Data** tab, select an attribute in the list and click **Apply**. A green check mark will appear next to the selected attribute. The Watflood Map will be updated in the view.

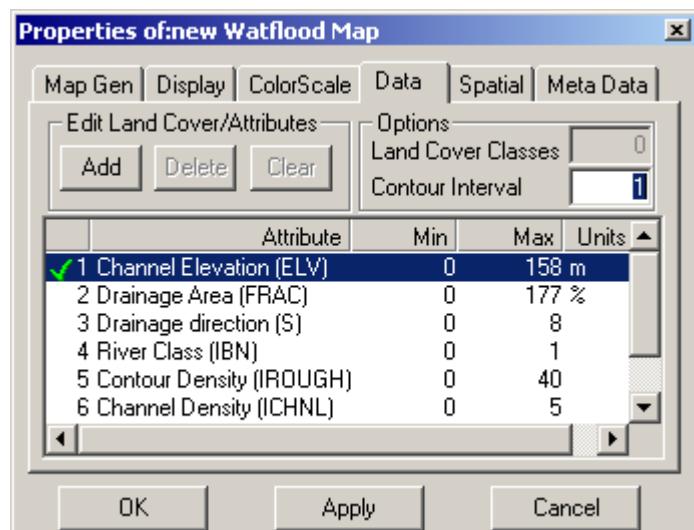


Figure 2.54: This tab determines which data attributes are displayed

4. The display and colour scale can be edited for each data attribute. See the sections Properties of Data Items and Colour Scale in the "Data Items" section.

To display all the data attributes for a single cell:

1. Select the Watflood Map.
2. Double-click on a cell of the Watflood Map. The cell will be highlighted in magenta and a box will appear with all the data attribute information for that cell.

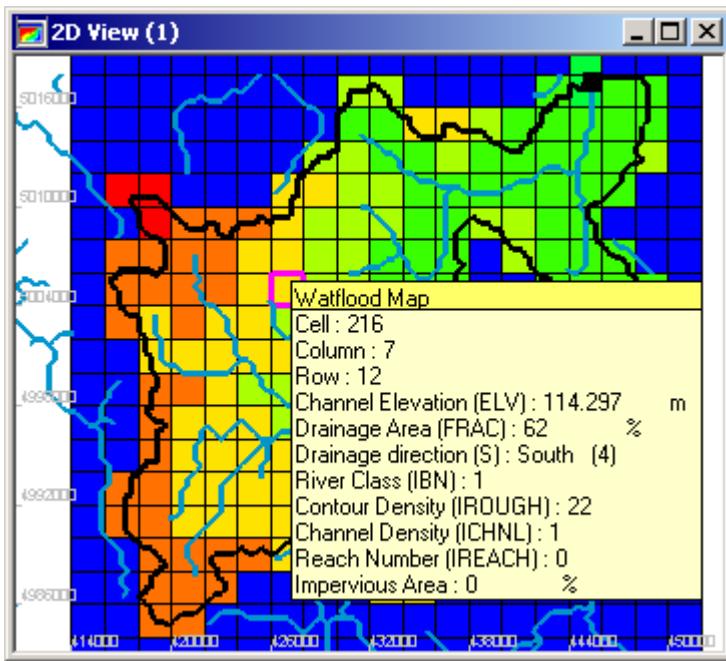


Figure 2.55: The popup shows all of the data information for the highlighted cell

2.3.1.5 Editing Watflood Map Data Attributes

The data attributes of a Watflood Map can be edited cell by cell. To check and edit the cells, the Watflood Map is best viewed by selecting the **Surface** style, and viewing the map in a 2D view.

To edit a single cell, select the cell and choose the **Edit** command from the shortcut menu. A dialog will appear, listing all of the data attributes. Simply click on the desired attribute and its value will become highlighted. Change the value and click **OK** or select another attribute to modify.

2.3.1.5.1 Adding Land Use Data Using Closed Polygons

Land use data attributes cannot be calculated from the DEM. Therefore, Green Kenue provides a tool to obtain land use information from GIS data.

The number of land uses described in the map file should correspond to the land-uses described in the WATFLOOD parameter file.

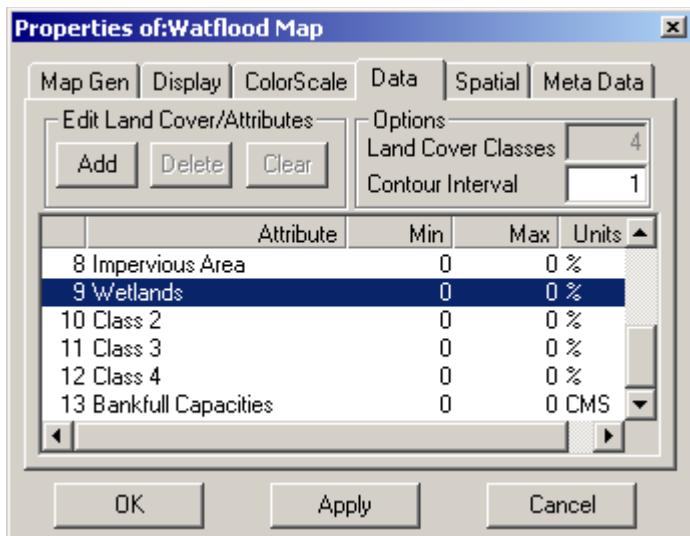


Figure 2.56: Use this dialog to add more land classes to the map

To add land class data:

1. In the **Data** tab of the Watflood Map's properties dialog, click the **Add** button. The **Add Attributes** dialog box will appear.

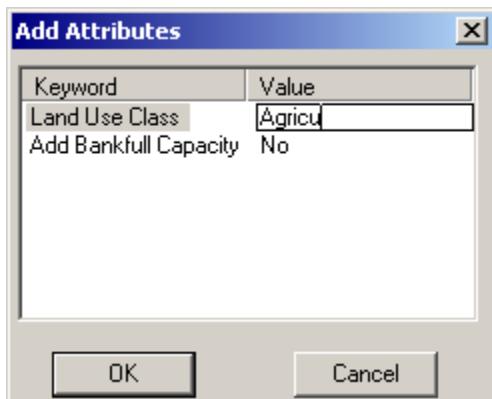


Figure 2.57: Use this dialog to enter the name of the new class

3. Click on a keyword to add a class to the map:

- **Land Use Class:** Enter the name of the new class in the **Value** column. You can change the name of the new class later, by double-clicking its name on the **Data** tab.
- **Add Bankfull Capacity:** Select **Yes** or **No** from the **Value** column. If you select yes, the **Bankfull Capacity** attribute will be added to the Watflood Map.

4. Click **OK**.

To map land use data:

1. Ensure that the Watershed object is a child of the Watflood Map. If it is not already associated, open the watershed object that was used to generate the map. In the WorkSpace, drag the watershed into the Watflood Map to associate the two. This will not change the

map's data attributes. In addition, ensure that the polygons or GIS files containing the land use data are available in the WorkSpace.

5. Select a land use class in the Data tab of the Watflood Map and press . A green check mark will appear to the left of the land class name.
6. Ensure that the Watflood Map is selected within the WorkSpace.
7. Select **Tools→Map Object**.
8. A dialog will appear listing the compatible objects. Select the object corresponding to the proper land use type and then select . When the land use data is applied, the grid cells will change colour according to the percentage of the cell's area that was covered by the land use polygons.
9. In the view, verify that the cells are coloured according to the percentage of the cell covered by the polygon(s). This can be done either by checking the colour against the colour scale, or by double clicking on a cell to bring up its attribute values.
10. Repeat steps 2 to 6 for each remaining land class.

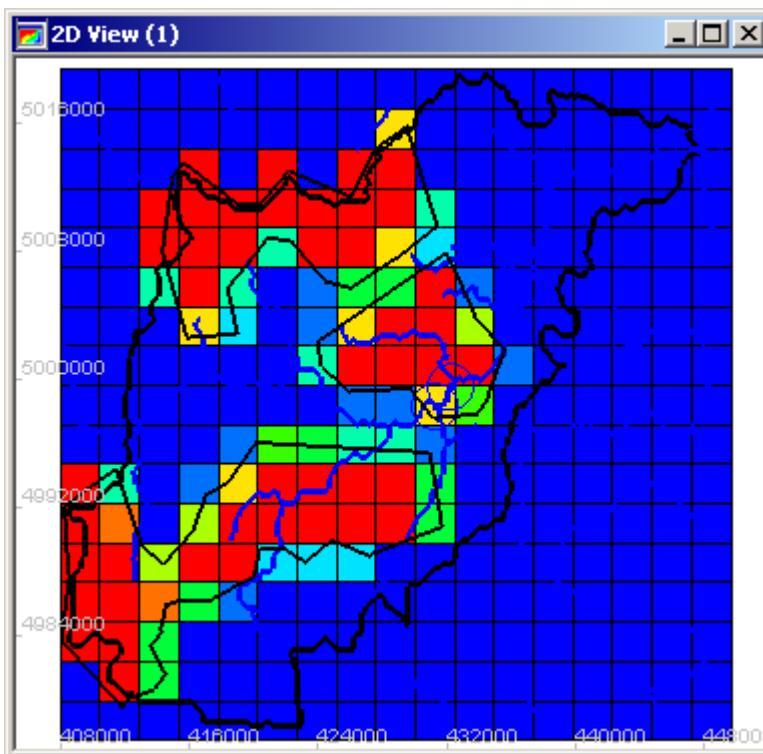


Figure 2.58: The colour of each cell indicates the percentage of its area belonging to a particular land class

The **Map Object** tool, described in the section "Mapping Objects" under Tools, on p. 113, can be used to apply one or more closed polygons defining land use to the Watflood map. These polygons can be from a GIS source, or can be created in Green Kenue using the polygon tool. See "Drawing Points", on p. 68 and "Drawing Lines and Closed Polylines" under Tools, on p. 69, for more information. The **Map Object** tool is designed to apply a value to a grid node, but for the Watflood Map, the percentage of the grid cell occupied by the mapped polygon is applied to the grid cell instead.

Points to remember when applying land use data to a Watflood Map:

- The land use polygons must be **closed**. Open polylines cannot be used.
- A data item that is being mapped to the Watflood Map may contain multiple polygons.
- Polygons can be applied to a land use class for which a polygon has already been applied. The percentages will be added to the existing percentages. Overlap of polygons between the first application and the second will not be accounted for.
- All of the polygons within a given GIS file must define the same land use. For example, all the polygons in the forest.shp GIS file must define forest land use, all the polygons in the urban.shp GIS file must define urban land use, and so on.
- Green Kenu cannot map complex polygons, such as nested polygons or polygons with holes. Using a GIS software package, split any complex polygons in the GIS file into simple polygons before applying the data.

2.3.1.5.2 Adding Land Use Data Using GeoTIFFs

Green Kenu provides an alternative tool to obtain land use information. Classes may be generated directly from one or more classified GeoTIFF images. The image(s) must be preprocessed so that only required landuse classes exist. See "Classification of a GeoTIFF Image" under How To - Hints and Tricks, on p. 130

To map land use data:

1. Ensure that a Watershed object is a child of the Watflood Map. If it is not already associated, open the Watershed object that was used to generate the Watflood Map. In the WorkSpace, drag the Watershed onto the Watflood Map to associate the two. This will not change the map's data attributes.
2. Import the GeoTIFF files with the preprocessed land use classification. Ensure that the full spatial extent of the basin(s) are covered by the GeoTIFF images.
3. Optionally, the preprocessed classification theme may be edited. On the Classes TAB of the GeoTIFF Properties dialog, enter the desired class names or select one of the predefined themes. Note: edits to the themes may be saved for reuse. The saved files are stored as ASCII .thm files in the bin\Templates\GeoTIFF directory.
4. Ensure that the Watflood Map is selected within the WorkSpace.
5. Select **Map Land Use from GeoTIFF(s)...** from the Watflood Map's shortcut menu.
6. A dialog will appear listing all available GeoTIFF objects. Select the appropriate GeoTIFF images and then select .

Each GeoTIFF image is examined and a unified list of land use classes is created. Next a pixel mask is created for each cell and the number of pixels of each land use class falling within the cell are counted and the integer percentages are assigned to the new Watflood Map land use classes. The resolution of the pixel mask is taken from the most detailed GeoTIFF image supplied.

Points to remember when creating land use classes from GeoTIFF images:

- When GeoTIFF images overlap, the value at a pixel is taken from the first image in the list.
- All GeoTIFF images must be classified using the same theme. However, not all classes have to be represented in all images.
- The supplied GeoTIFF images must fully cover the Watershed basin(s) used to create the Watflood Map.
- The supplied GeoTIFF images may have differing resolutions. Green Kenu uses the pixel size of the highest resolution image supplied when counting pixels.

2.3.1.5.3 Editing Land Use Data

Land use data can be edited in the same way as other data attributes of the Watflood Map file. See "Editing Watflood Map Data Attributes", on p. 183, for more information.

2.3.1.5.4 Resetting a Land Use Class

To reset a land use class:

1. Select the land class attribute on the Data tab of the Watflood Map's properties dialog. Click  and ensure that there is a green checkmark to the left of the attribute you wish to clear.
2. Click . The selected land class will now have a value of 0% for each cell.

2.3.1.6 Saving the Watflood Map

The Watflood Map can be saved as a *.map file or as a *.r2c file. Select the Watflood Map object, and choose **File→Save** or **File→Save Copy As....**. See "Supported Foreign File Types [Green Kenu]", on p. 311, for information about the *.map file format. Note: The Watflood Map saved as an EnSim *.r2c file can be recognized as input by newer versions of the Watflood Model.

2.3.2 Importing WATFLOOD Files

WATFLOOD files (other than the Watflood Map and the Watflood Output, which are loaded through the open menu) can be imported into Green Kenu. With the exception of the Watflood Event file, the files may be viewed but cannot be edited or saved in their native formats. Refer to the section on "Watflood Event File Properties", on p. 188, for details on the Event file. **Note:** Imported Watflood grid files such as *.tem and *.met files can be saved as EnSim *.r2c files which can be recognized by newer versions of the WATFLOOD model.

To add a Watflood file, select **File→Import→Watflood Files...**. Another menu will appear showing the Watflood files structure.

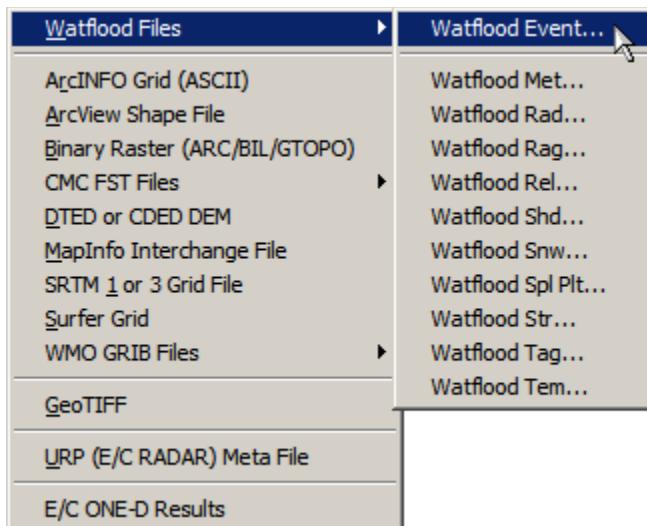


Figure 2.59: This menu is used to import a WATFLOOD file

Once you have selected one of the file types, the **Open** dialog will appear. Locate and select the desired file and click **Open**.

2.3.2.1 Watflood Event File Properties

Upon opening a Watflood Event file (*.evt), all associated data files will be loaded into the WorkSpace. Currently, an Event file can only be edited but not created within EnSim.

Unlike most data objects, Event files have only one tab in their Properties dialog: **Data**.

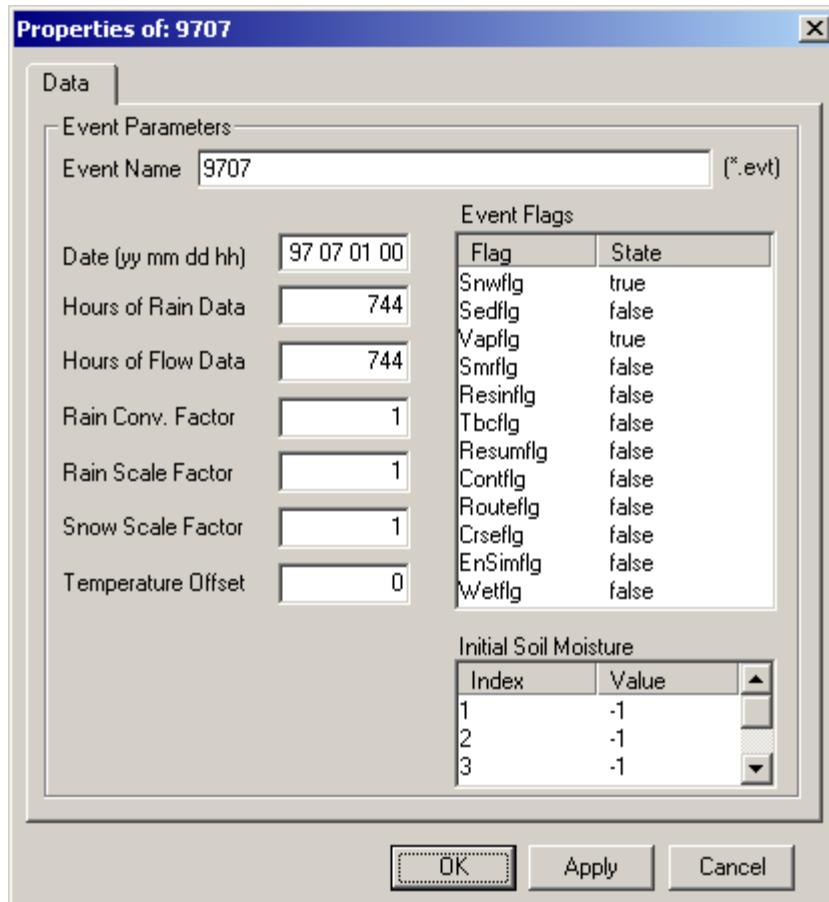


Figure 2.60: This dialog allows you to edit the WATFLOOD Event file

Refer to your WATFLOOD User's Manual for details on the Event file.

The parameters **Event Name**, **Data**, **Hours of Rain**, **Hours of Flow Data**, **Rain Conv. Factor**, **Rain Scale Factor**, **Snow Scale Factor**, and **Temperature Offset** may be changed to reflect the desired values. To change the state of the Event flags, click on either of the **Flag** or the **State** to toggle the flag. To change the **Initial Soil Moisture** values in the parameter file, click on the index number or the value to edit the value.

To save changes to the Event file:

1. With the Event file selected in the WorkSpace, select **File**→**Save Copy As...**
2. Enter a file name and click **Save**. If you changed the **Event Name** parameter in the **Properties** dialog, the new name will be listed here as the default file name.

2.3.3 WATFLOOD Output

Watflood Binary Output (*.wfo) files can be opened in Green Kenu. A Binary Output file will appear in the WorkSpace with various components displayed as children.

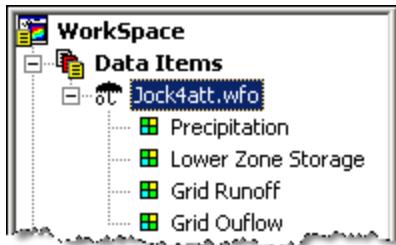


Figure 2.61: A Watflood Binary Output file

The Watflood Output file may have many more components than this example. Possible components include Temperature, Precipitation, Grid Runoff, Grid Outflow, and Lower Zone Storage, as well as Depression Storage, Depression Storage (Snow), Snow Water Equivalent, Snow Covered Area, and Upper Zone Storage for each land use class.

Each component of the Watflood output file can be displayed in a 2D or 3D view. Multiple components may be overlaid in a view.

Components of the output file, such as Grid Outflow, can be saved independently as single-frame ASCII 2D rectangular cell files or as multi-frame binary 2D rectangular cell files. The single-frame option will save only the current frame of the component file. The multi-frame option will save all the frames in the time series. See "2D Rectangular Cell Grids [r2c]", on p. 300, for more information about 2D rectangular cell and "Supported Foreign File Types [Green Kenu]", on p. 311 for more information on Watflood Binary Output files.

2.3.4 Bankfull Animation

Bankfull animation allows you to visualise the streams and their potential for flooding during a particular event. Based on the modelled flow, the network of streams, and the bankfull capacities of the main stream in each cell of the Watflood Map, the percentage flows of the streams are displayed at each time step. A percentage greater than 100% indicates potential flooding of the stream. When viewing the bankfull animation, you can change the colour scale to display flooding in an alternate colour.

Over the course of the animation, the stream segment within a cell changes colour according to the percentage bankfull flow at each time step. Time series can be extracted from a selected stream segment, showing the changes in bankfull flow over time. Time series are displayed in a 1D view.

To create a bankfull animation:

Note: A bankfull animation can only be created after a watershed has been modelled with WATFLOOD.

1. Open the watershed object (*.wsd).
2. Change the display of the channels to the desired density. That is, show as many of the minor channels as desired.
3. Save the channels as an independent network file (*.n3s).

4. Open the Watflood Output file (*.wfo) and save the grid outflow component as a binary multi-frame file.
5. Select **File→New→Watflood Bankfull...**. The **New Bankfull Animation** dialog will appear.

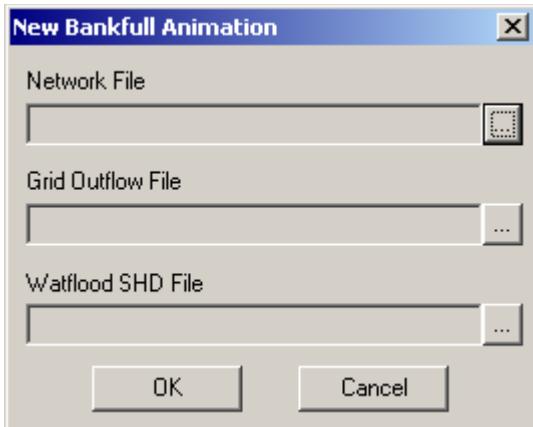


Figure 2.62: This dialog is used to set up a Bankfull animation

6. Specify the network, grid outflow, and shed files in the dialog. Click **...** to browse. When you're finished, click **OK**.
7. A **Save As...** dialog will prompt you to save the bankfull animation file as a binary multi-frame file.
8. When you click **Save**, the bankfull values will be calculated, saved, and loaded into the WorkSpace.

Bankfull animation files can be displayed and animated in Green Kenue in the same manner as other time varying data objects.

3 ENVIRONMENTAL DATABASES

3.1 HYDAT DATABASE

3.1.1 Introduction

Surface water quality data has been collected and archived in Canada since the middle of the last century. Beginning in 1908 this data has been published in a variety of printed formats. Since 1991, the meteorological Service of Canada has published most of this data on CD-ROM.

This National HYDAT CD-ROM, produced each year, provides rapid access to a stand-alone version of the National Water Data Archive. This large database contains daily, monthly, and/or instantaneous information for streamflow, water level, suspended sediment concentration, sediment particle size, and sediment load data for over 2900 active stations and some 5100 discontinued sites across Canada.

Green Kenu provides a graphical interface to the HYDAT database with which you can query, display, and analyze the data associated with each station.

3.1.2 Accessing the Database

The HYDAT Database is available in two forms: the HYDAT CD and the HYDAT Access database, or .MDB file.

3.1.2.1 The HYDAT Database CD

The database resides in two separate directories on the CD. These directories are:

- **CD:\HSIS6\DATA** - This directory contains subdirectories organized by province, which in turn contain HYDAT station index files arranged by major watersheds.
- **CD:\HYDAT6** - This directory contains 100 subdirectories, which in turn contain the station data files.

The HYDAT data can be accessed directory from the CD, or the directories can be copied to a local hard drive, or to a network drive.

Green Kenu first looks for the \HSIS6\Data and \HYDAT6 directories below the location of the GreenKenu.exe file (e.g. C:\Program Files\CHC\GreenKenu\HSIS6); then if not found, the application searches all drive letters from **C:** to **Z:** and selects the first location containing the \HSIS6\DATA and \HYDAT6 directories as the locations of the database.

To access the HYDAT database CD:

1. Select **File**→**Environmental Data**→**Open HYDAT** from the menu bar.

2. The HYDAT stations can be accessed for all of Canada, or by individual province. Select one of the choices shown in Figure 3.1.

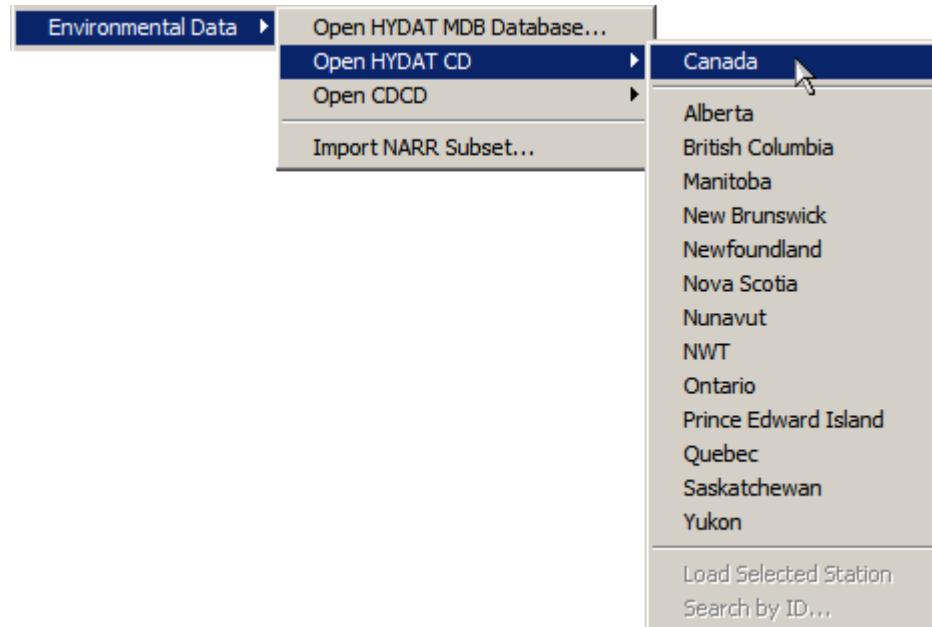


Figure 3.1: This menu is used to access the HYDAT database on CD

Once the selected data has been loaded, the corresponding province becomes greyed out. Additional selections, including the Canada option, add stations to the HYDAT object. The HYDAT object is a multi-attribute point set, which can be displayed in a 2D view.



Figure 3.2: The HYDAT object appears in the WorkSpace

3.1.2.2 The HYDAT MDB Database

The HYDAT MDB Database is contained within a single file named **Hydat.mdb**. This version of the HYDAT database includes some fields that aren't available in the CD version. The HYDAT MDB Database can be obtained from Environment Canada.

To access the HYDAT MDB Database, you must have Microsoft Access 2007, or the Microsoft Access 2007 Runtime Library installed on your computer. The latter can be downloaded from the Microsoft website. Note that if you are using a 64-bit Operating System, you'll need to install the 64-bit version of Microsoft Access 2007 or the Microsoft Access 2007 Runtime Library.

To open the HYDAT MDB Database:

1. From the menu bar, select **File**→**Environmental Data**→**Open HYDAT MDB Database...**

2. Browse to the location of the Hydat.mdb file and click **Open**. The window opens to the location of the Green Kenu program, e.g., C:\Program Files\CHC\GreenKenu, by default.

The HYDAT MDB Database is a multiattribute point set which can be displayed in a 2D view. It contains points for each of the HYDAT measuring stations across Canada.

3.1.3 Accessing Station Details

Once the HYDAT object is in the WorkSpace, you can access detailed information in several ways.

To access a selected station:

1. Click on a station within a View, as shown in Figure 3.3. All attributes of the selected station will be displayed.

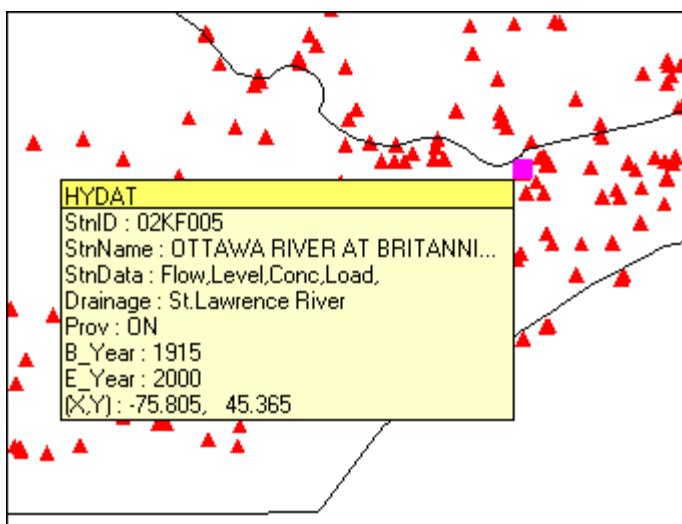


Figure 3.3: The attributes of a station can be accessed in a view window

2. Right-click on the selected object and select **Load Selected** from the shortcut menu, or select **File→Environmental Data→Open HYDAT→Load Selected** from the menu bar. Station details, as well as associated time series, are then shown in the WorkSpace as children of the selected station.

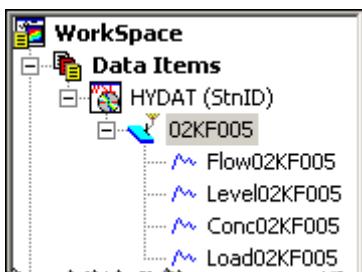


Figure 3.4: Data from a specific station are shown as children of that station

To access a station by ID:

1. Select **File**→**Environmental Data**→**Open HYDAT**→**Search by ID...** from the menu bar.

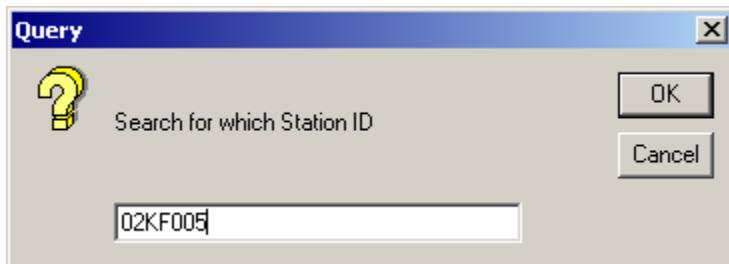


Figure 3.5: This dialog allows you to search for a HYDAT station by ID

2. Enter the HYDAT ID number in the dialog and click **OK**. All HYDAT ID numbers are uppercase.

Station details and associated time series will be shown in the WorkSpace as children of the selected station, as shown in Figure 3.4.

3.1.4 Filtering Station Details

The contents of the HYDAT MDB Database can be filtered to show only a subset of available stations. Filtering the database restricts the displayed records both within a View and within the Attribute Table for the HYDAT data object.

To filter the HYDAT MDB Database:

1. Within the WorkSpace, right-click on the HYDAT MDB Database object, **HydatStations (Station)** and select **Show Attribute Table (Filter)**... from the shortcut menu.

The **HydatStations Filter** table will appear in the View area.

Station	Station Name	HydStatus	Prov	Latitude	Longitude	Drai
01AA002	DAAQUAM (RIVIERE) EN AVAL DE LA RIVIERE SHIDGEL	Discontinued	QC	46.5575	-70.08111	
01AD001	MADAWASKA (RIVIERE) EN AVAL DU BARRAGE TEMISCOUATA	Discontinued	QC	47.54833	-68.63639	
01AD002	SAINT JOHN RIVER AT FORT KENT	Active	ME	47.25806	-68.59583	
01AD003	ST. FRANCIS RIVER AT OUTLET OF GLASIER LAKE	Active	NB	47.20661	-68.95694	
01AD004	SAINT JOHN RIVER AT EDMUNDSTON	Active	NB	47.36078	-68.32489	
01AD005	MADAWASKA (RIVIERE) AU RESERVOIR TEMISCOUATA	Active	QC	47.57056	-68.64306	
01AD008	LONG (LAC) PRES DE LES ETROITS	Discontinued	QC	47.39083	-68.89833	
01AD009	CABANO (RIVIERE) AU LAC LONG	Discontinued	QC	47.46444	-69.00194	
01AD012	SAINT-FRANCOIS (RIVIERE) EN AVAL DU LAC SAINT-FRAN...	Discontinued	QC	47.72472	-69.28333	
01AD013	SAINT-FRANCOIS (RIVIERE) EN AVAL DU LAC SAINT-FRAN...	Discontinued	QC	47.71917	-69.28389	
01AD014	POUINECAMOON (LAC) A ESCOULOT	Discontinued	QC	47.46	-69.28555	

Figure 3.6: This HYDAT MDB Attribute Table is currently unfiltered

2. Enter or select the restrictions from each of the available fields that apply to the subset of stations that you'd like to examine. Note that some fields may be empty for some records; searching for any specific value will not return records that contain no value for that field.

- **Station Number:** This is a seven-character alphanumeric code that uniquely identifies each station. The code follows the format **##XX###**, where # represents a digit and X is a letter. The search string matches only the beginning of each code and does not use wildcards.
- **Station Name:** This search field looks for the appearance of the provided text anywhere within the Station Name field of the database. For example, searching for the word "Brook" will restrict the database to recording stations whose names contain the word "Brook", as well as those containing "Sherbrooke", "Cranbrook", or "Brookmere".
- **Province:** This drop-down list allows you to search for recording stations located within a single province. Note that several U.S. border states also contain recording stations and can be searched with this field. The complete list consists of:
 - **AB:** Alberta
 - **BC:** British Columbia
 - **MB:** Manitoba
 - **NB:** New Brunswick
 - **NL:** Newfoundland and Labrador
 - **NS:** Nova Scotia
 - **NT:** Northwest Territories
 - **NU:** Nunavut
 - **ON:** Ontario
 - **PE:** Prince Edward Island
 - **QC:** Quebec
 - **SK:** Saskatchewan
 - **YT:** Yukon Territory
 - **ME:** Maine
 - **MN:** Minnesota
 - **MT:** Montana
 - **ND:** North Dakota
 - **AK:** Alaska
 - **WA:** Washington
 - **ID:** Idaho
- **Hyd Status:** This field may be either **Active** or **Discontinued**.

- **Regulation:** This field may be either **Regulated** or **Natural**.
- **Oper Sched:** This field, short for Operating Schedule, may be either **Continuous**, **Miscellaneous**, or **Seasonal**.
- **Data Period:** This field allows you to set a date range. Applying the filter will return all records that contain at least some data falling within the given range.
- **Total Years:** This field consists of a drop-down list and a text box. From the list, select \leq (less than or equal to) or \geq (greater than or equal to). In the text box, enter a number. The filter will return all records that contain data from at most or at least the given number of years.
- **Drainage Area:** This field consists of a drop-down list and a text box. From the list, select \leq (less than or equal to) or \geq (greater than or equal to). In the text box, enter a number. The filter will return all records for stations that have at most or at least the given drainage area.
- **Has Flow:** This checkbox allows you to show only records that contain flow data.
- **Has Level:** This checkbox allows you to show only records that contain level data.
- **Has Sed:** This checkbox allows you to show only records that contain sedimentation data.
- **Is Real Time:** This checkbox allows you to show only records that contain real-time data, as opposed to periodic measurements.
- **Is RHBN:** This checkbox allows you to show only records that are part of the reference hydrometric basin network.

3. Click **Apply Filter** to restrict the displayed records to the selected criteria. To undo the restriction, clear all search fields, and show all available records, click **Reset Filter**.

Note: You can sort the list of records within the attribute table by clicking on any of the column headers. Clicking on a header again will reverse the sort order. By default, the records are sorted in ascending order by station number.

3.1.5 Properties of a HYDAT Station

The HYDAT station properties are:

- **Details**
- **HYDEX**
- **Meta Data**

3.1.5.1 Details

The **Details** tab displays most of the pertinent data linked with this station. There are three sections to this tab.

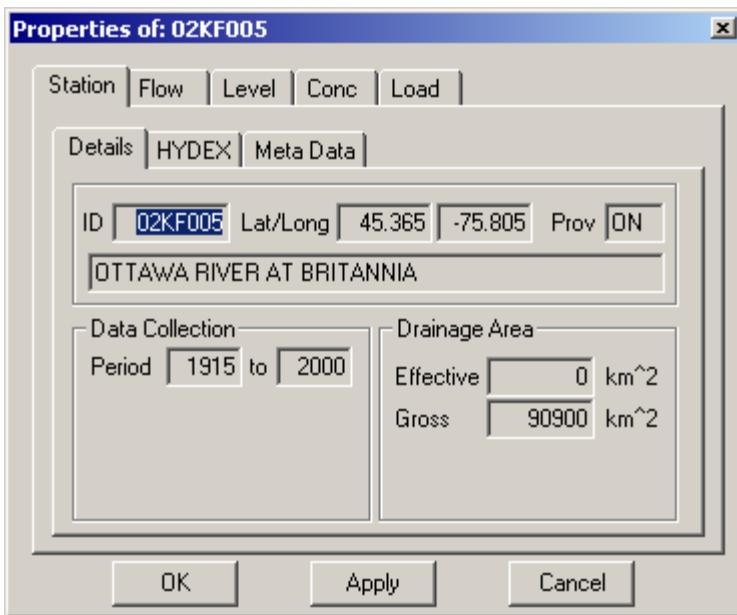


Figure 3.7: The details of a HYDAT station cannot be edited directly

- **Identification:** This section details the location and ID number of the station.
 - **ID:** This is the HYDAT ID number. There is a unique ID number for each station.
 - **Lat/Long:** This is the latitude and longitude of the station, in decimal degrees.
 - **Prov.:** This is the province in which the station is located.
 - **Name (not labelled):** This is the official name of the station.
- **Data Collection:**
 - **Period:** This identifies the first and last years during which data was collected at this station.
- **Drainage Area:**
 - **Effective:** This is the effective drainage area of the watershed, in km².
 - **Gross:** This is the gross drainage area of the watershed, in km².

3.1.5.2 HYDEX

This tab displays HYDEX information. This tab also has three sections. Refer to the HYDAT documentation for complete information on the HYDEX.

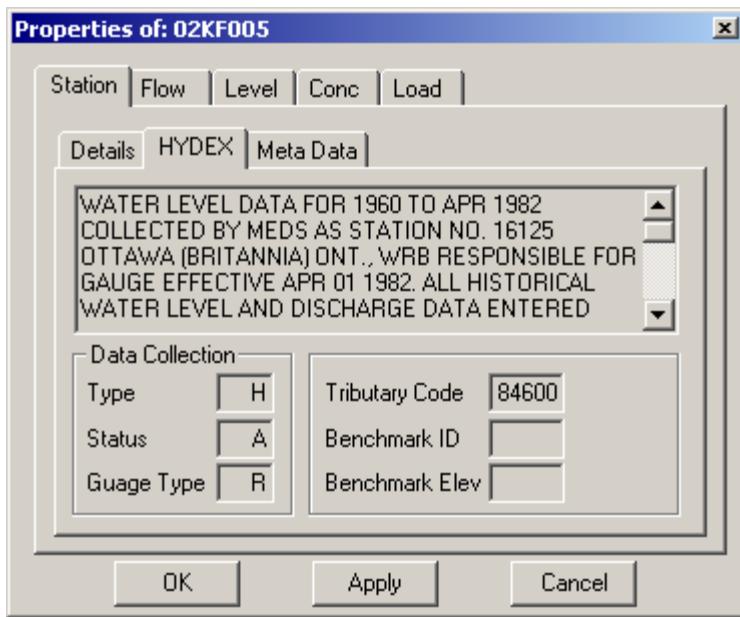


Figure 3.8: This tab displays additional HYDEX information.

3.1.5.3 Meta Data

Additional HYDEX information can be found on the Meta Data tab. See "Meta Data" under Properties of Data Items, on p. 31, for more information.

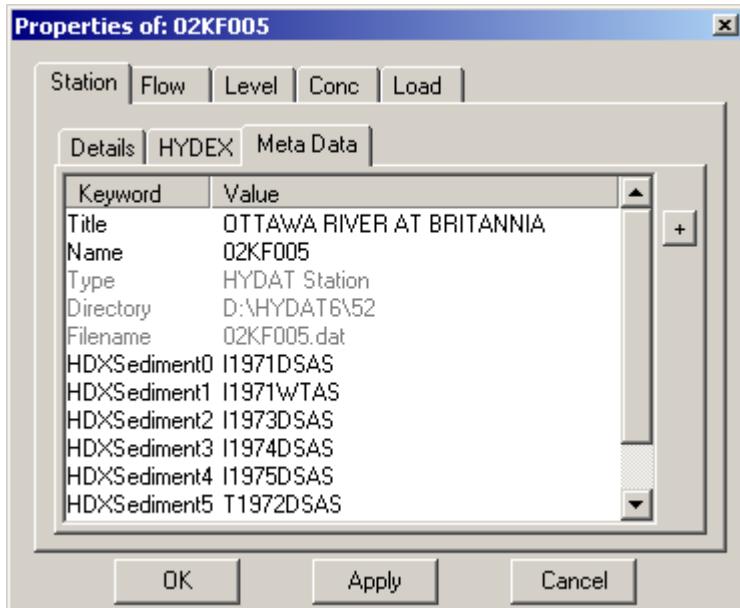


Figure 3.9: The Meta Data tab displays data about the station files, as opposed to data about the station

3.1.6 Properties of Associated Time Series

There are four possible time series that can be linked to a Hydat station. These include Flow, Level, Conc, and Load. For each time series linked to the station, a tab will appear in the properties dialog. These time series are similar to other EnSim time series, with the addition of the **Subset** tab.

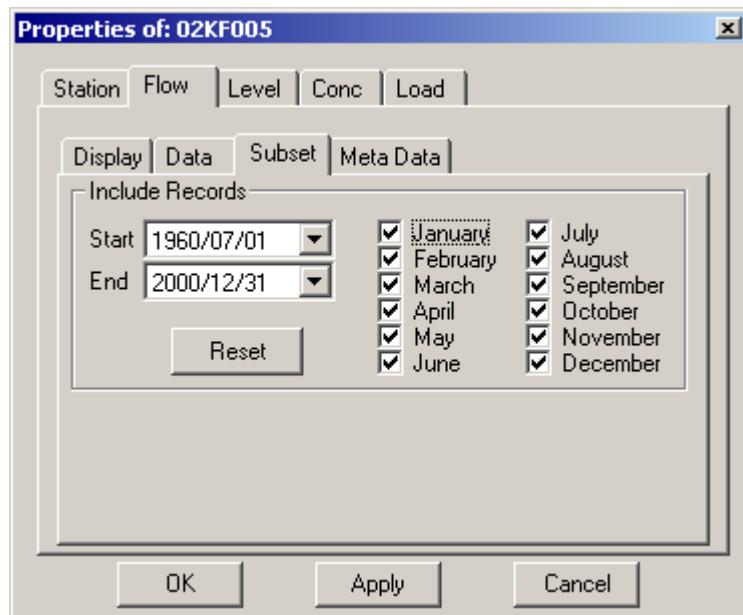


Figure 3.10: Each of the four time series has a Subset tab

3.1.6.1 Subset

The **Subset** tab allows you to adjust the temporal range of data shown in the 1D view. Initially, the entire data set will be shown.

To create a temporal subset:

1. Click on either the Start or End data boxes. You can adjust the date by typing the new data, or by clicking on the button. If the button is selected, a calendar will appear.

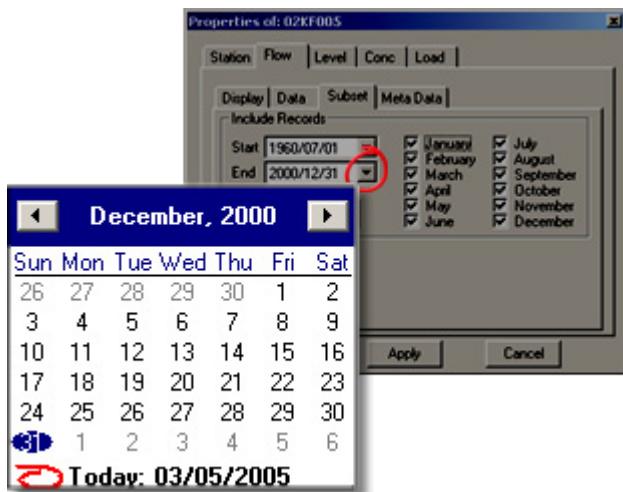


Figure 3.11: This calendar is used to change the start and end dates of a temporal subset

2. Click on the date to which you would like to change the start or end time.
 - Click on **◀** or **▶** to adjust the month backward or forward, or click on the calendar title to select a specific month.
 - Click on the year and use the up and down arrows to change the year.
3. To remove months from the data set, click on the check box next to the month name on the **Subset** tab.
4. Click the **Reset** button to include the entire data set.
5. Select the **Apply** button to apply your changes. If you have selected any dates outside the range of data for the station, the date will reset to the maximum for that parameter.

3.2 CDCD DATABASE

3.2.1 Introduction

The Canadian Daily Climate Data (CDCD) contains daily temperature, precipitation and snow on the ground data for almost 8000 locations in Canada.

Green Kenuue provides a graphical interface to the CDCD database with which you can query, display, and analyze the data associated with each station.

3.2.2 Accessing the Database

The CDCD data can be accessed directory from the DVD, or the directories can be copied to a local hard drive, or to a network drive.

Green Kenuue first looks for the **\CDCD\1** directory below the location of the GreenKenuue.exe file (e.g. C:\Program Files\CHC\GreenKenuue\CDCD\1); then if not found, the application searches all drive letters from **C:** to **Z:** and selects the first location containing the **\CDCD\1** directory as the locations of the database. Note: The files DATA.101 and INDEX.101 must be present in the **\CDCD\1** directory.

To access the CDCD database:

1. Select **File**→**Environmental Data**→**Open CDCD** from the menu bar.
2. The CDCD stations can be accessed for all of Canada, or by individual region. Select one of the choices shown in Figure 3.12.

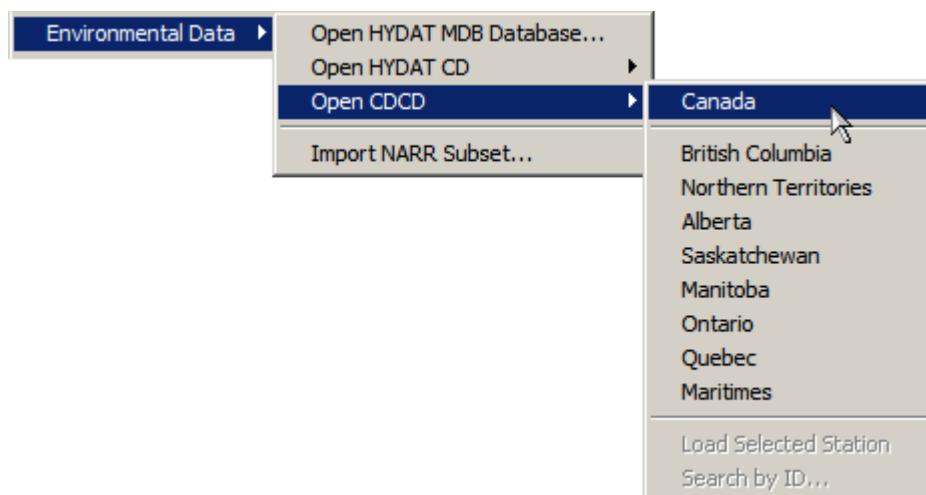


Figure 3.12: This menu is used to access the CDCD database

Once the selected data has been loaded, the corresponding region becomes greyed out. Additional selections, including the Canada option, add stations to the CDCD object. The CDCD object is a multi-attribute point set, which can be displayed in a 2D view.



Figure 3.13: The CDCD object appears in the WorkSpace

3.2.3 Accessing Station Details

Once the CDCD object is in the WorkSpace, you can access detailed information in several ways.

To access a selected station:

1. Click on a station within a View, as shown in Figure 3.3. All attributes of the selected station will be displayed.

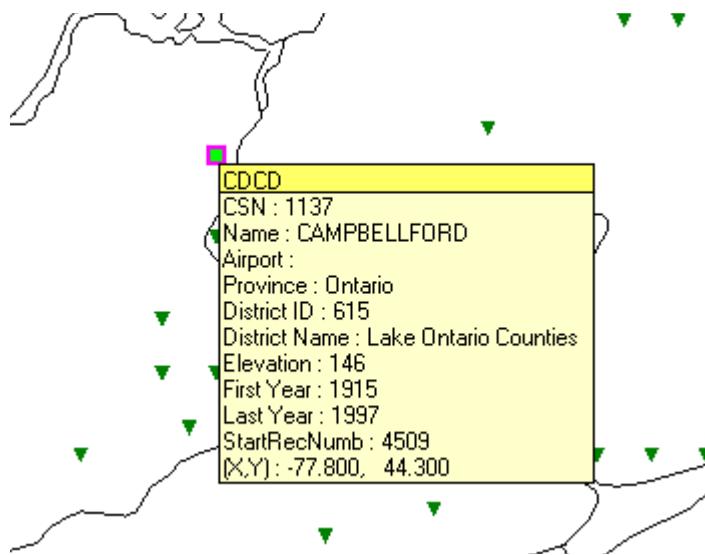


Figure 3.14: The attributes of a station can be accessed in a view window

2. Right-click on the selected object and select **Load Selected** from the shortcut menu, or select **File→Environmental Data→Open CDCD→Load Selected** from the menu bar. Station details, as well as associated time series, are then shown in the WorkSpace as children of the selected station.

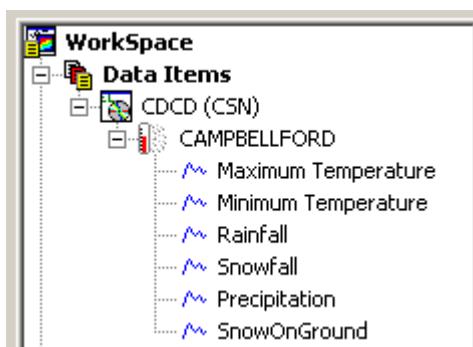


Figure 3.15: Data from a specific station are shown as children of that station

To access a station by ID:

1. Select **File**→**Environmental Data**→**Open CDCD**→**Search by ID** from the menu bar.

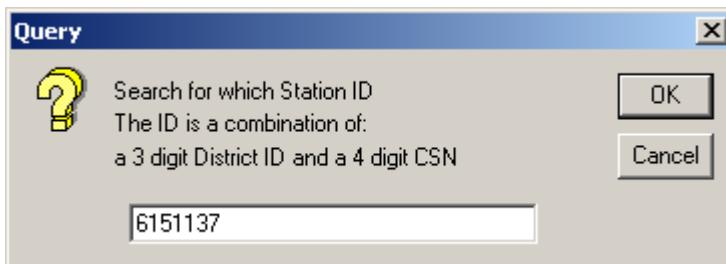


Figure 3.16: This dialog allows you to search for a CDCD station by ID

2. Enter the CDCD ID number in the dialog and click **OK**.

Station details and associated time series will be shown in the WorkSpace as children of the selected station, as shown in Figure 3.15.

3.2.4 Properties of a CDCD Station

The CDCD station properties are:

- **Details**
- **Meta Data**

3.2.4.1 Details

The **Details** tab displayed most of the pertinent data linked with this station. There are two sections to this tab.

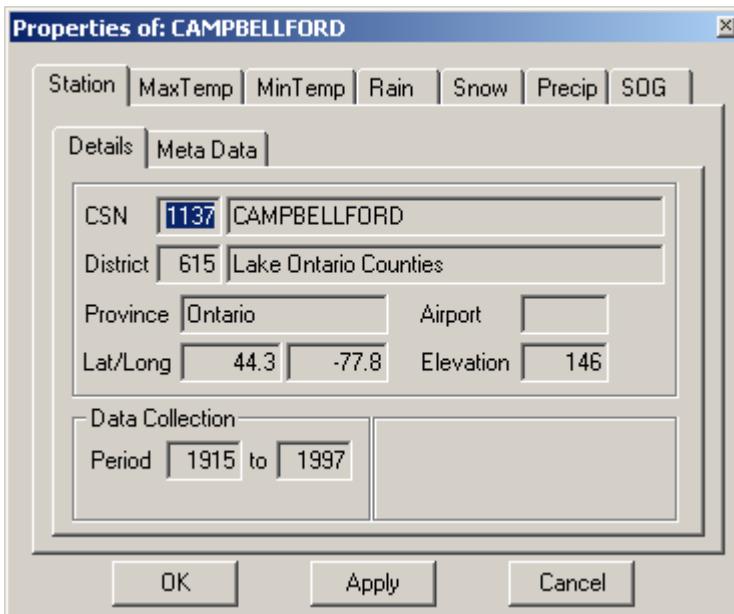


Figure 3.17: The details of a CDCD station cannot be edited directly

- **Identification:** This section details the location and ID number of the CDCC station.
 - **CSN:** This is the ID and name of the station.
 - **District:** This is the district ID and district name in which the station is situated.
 - **Province:** This is the province in which the station is situated.
 - **Lat/Long:** This is the latitude and longitude of the station, in decimal degrees.
 - **Airport:** This is the Airport ID (applicable only if the station is an airport).
 - **Elevation:** This is the elevation of the station.
- **Data Collection:**
 - **Period:** This details the years during which data was collected at this station.

3.2.4.2 Meta Data

See "Meta Data" under Properties of Data Items, on p. 31, for more information.

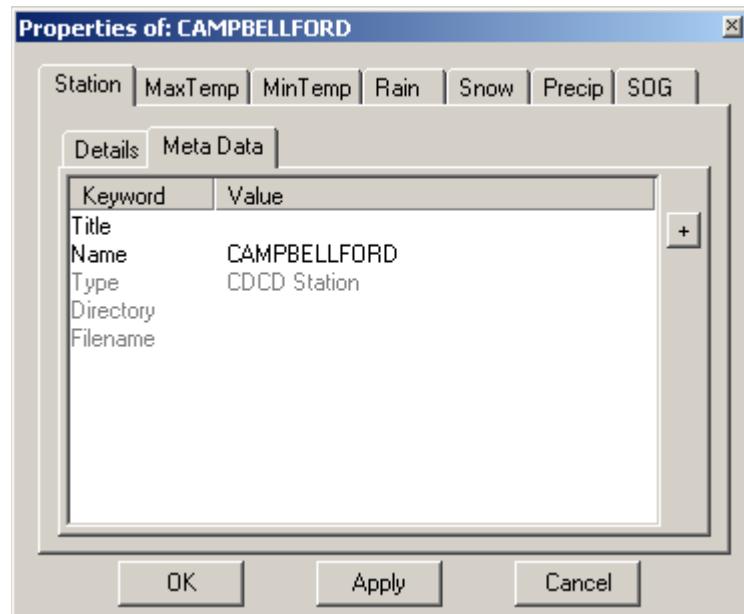


Figure 3.18: The Meta Data tab of the CDCC station

3.2.5 Properties of Associated Time Series

There are six possible time series that can be linked to a CDCC station. These include MaxTemp, MinTemp, Rain, Snow, Precip, and SOG (snow on ground). For each time series linked to the station, a tab will appear in the properties dialog. These time series are similar to other EnSim time series, with the addition of the **Subset** tab.

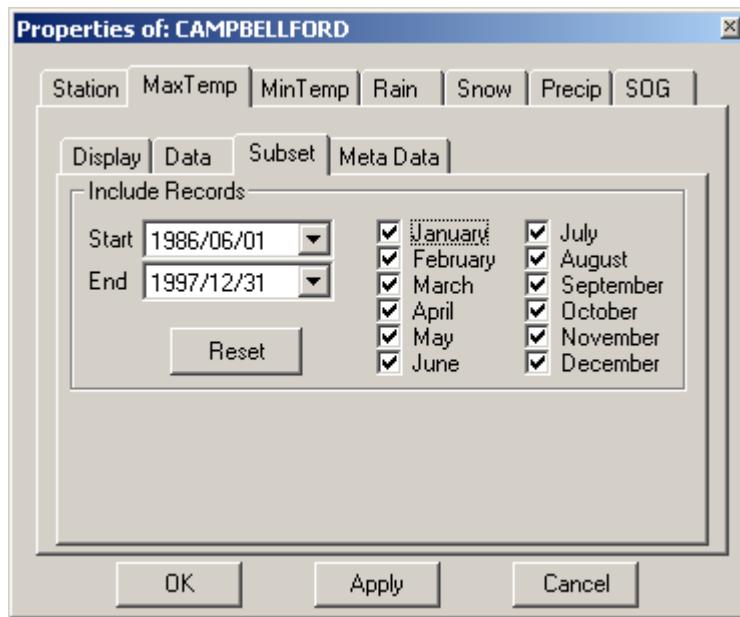


Figure 3.19: Each of the six time series has a Subset tab

3.2.5.1 Subset

The **Subset** tab allows you to adjust the temporal range of data shown in the 1D view. Initially, the entire data set will be shown.

To create a temporal subset:

1. Click on either the Start or End data boxes. You can adjust the date by typing the new data, or by clicking on the button. If the button is selected, a calendar will appear.

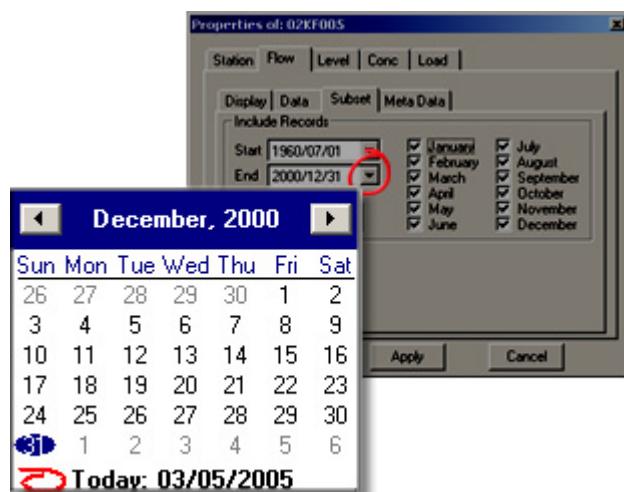


Figure 3.20: This calendar is used to change the start and end dates of a temporal subset

2. Click on the date to which you would like to change the start or end time.

- Click on or to adjust the month backward or forward, or click on the calendar title to select a specific month.

- Click on the year and use the up and down arrows to change the year.

3. To remove months from the data set, click on the check box next to the month name on the **Subset** tab.
4. Click the  button to include the entire data set.
5. Select the  button to apply your changes. If you have selected any dates outside the range of data for the station, the date will reset to the maximum for that parameter.

3.3 NARR DATABASE

3.3.1 Introduction

The North American Regional Reanalysis dataset is a long term, consistent, high resolution database of approximately 180 climatological parameters that cover the North American continent. It was developed at the National Center for Environmental Prediction.

This dataset has a spatial resolution of 32 km on a 349 by 277 grid and a temporal resolution of 3 hours between January 1979 and the present.

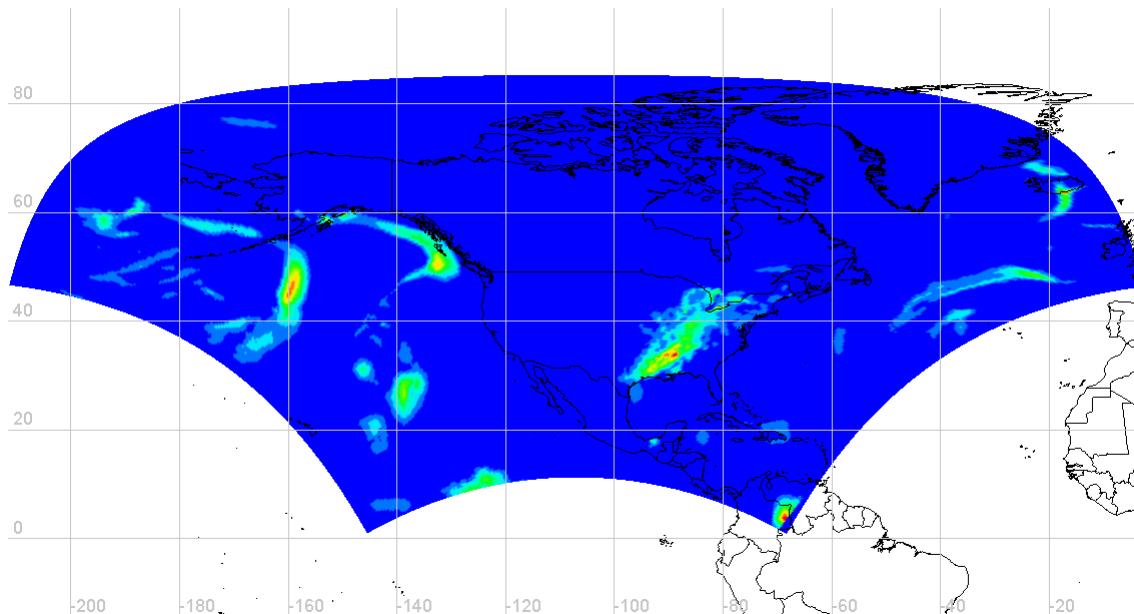


Figure 3.21: NARR data coverage

More information can be found at <http://www.emc.ncep.noaa.gov/mmb/rreanal/>.

3.3.2 Downloading the NARR Data

NCEP provides a download service where users can extract individual parameters from the NARR database via a set of perl scripts. Perl is open source and freely available. A standard distribution called ActivePerl can be downloaded for free from www.activestate.com.

For the sake of convenience these scripts have been included in the Green Kenue distribution and can be found in the directory `{INSTALL_DIR}\NARR`. They files include:

- `get-httpssubset.pl`
- `get_inv.pl`
- `get_grib.pl`
- `get_gfs.pl`

One executable file is also provided:

- curl.exe

The perl script files may have to be edited manually to ensure that the directory paths are correct. For example, the following lines at the top of the get-httpssubset.pl file may have to be edited if get_inv.pl and get_grib.pl are not in the specified directory:

```
$get_inv = "C:/NARR/get_inv.pl";  
$get_grib = "C:/NARR/get_grib.pl";
```

Another edit may have to be made in both the get_gfs.pl and get_inv.pl files:

```
$curl="C:/NARR/curl";
```

The main script, get-httpssubset.pl, is invoked with a number of parameters:

```
perl get-httpssubset.pl {startDate} {endDate} {parameter} {level} {dir} {database}
```

For example, the following script:

```
perl get-httpssubset.pl 1979010100 2006123121 TMP 2_m ./TMP_2 narr-a
```

downloads the variable TMP (temperature) for 2m AGL between midnight Jan. 1, 1979 and 2100 hrs Dec. 31, 2006 into the local subdirectory TMP_2. The directory TMP_2 will be further subdivided into directories for every month named using the format {YYYYMM}.

e.g. directory: .\TMP_2m\197901 for January 1979

e.g. filename: **narr-a_221_20061231_2100_000.sub.grb** for Dec. 12, 2006 at 2100 hours

The downloaded files are in GRIB format

3.3.3 Accessing the NARR Variables

Green Kenua provides an import tool for this dataset. This tool allows you to create .r2s or .r2c files from a directory containing a single NARR variable.

To import the NARR data:

1. Select **File→Environmental Data→Import NARR Subset...** from the menu bar. The following dialog will appear.

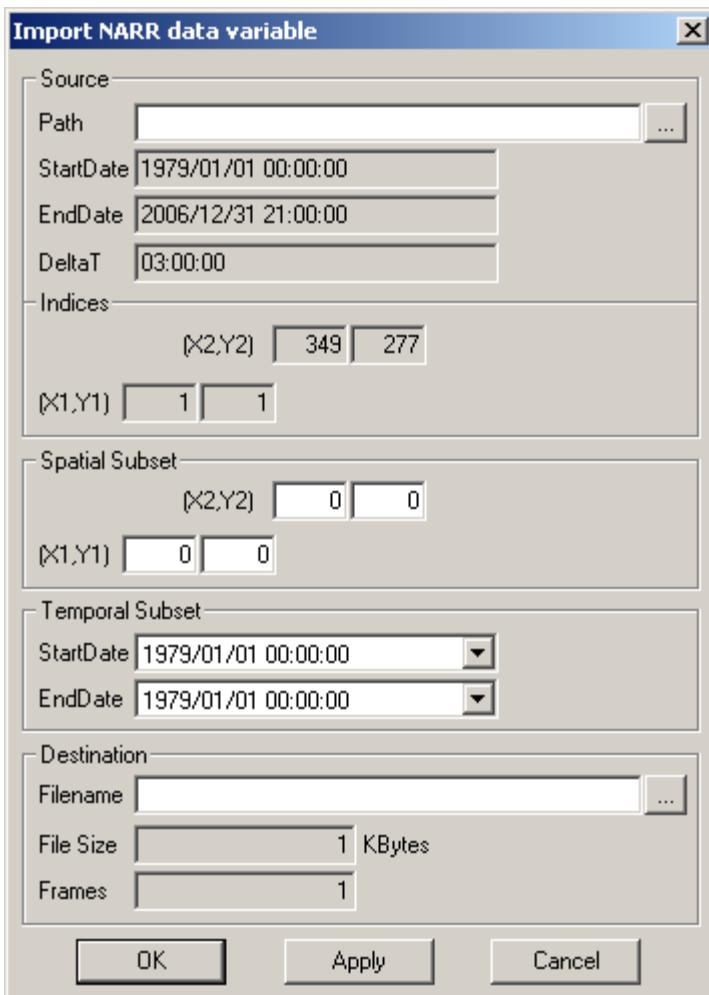


Figure 3.22: The Import NARR data variable dialog

2. Select the directory **Path** for the source NARR data by clicking on the button to browse to the directory, or by typing the path directly into the window.

Note: When choosing the Source Path, select the directory representing the variable (e.g. TMP_2m), not the subdirectory for a specific month (e.g. 197901).

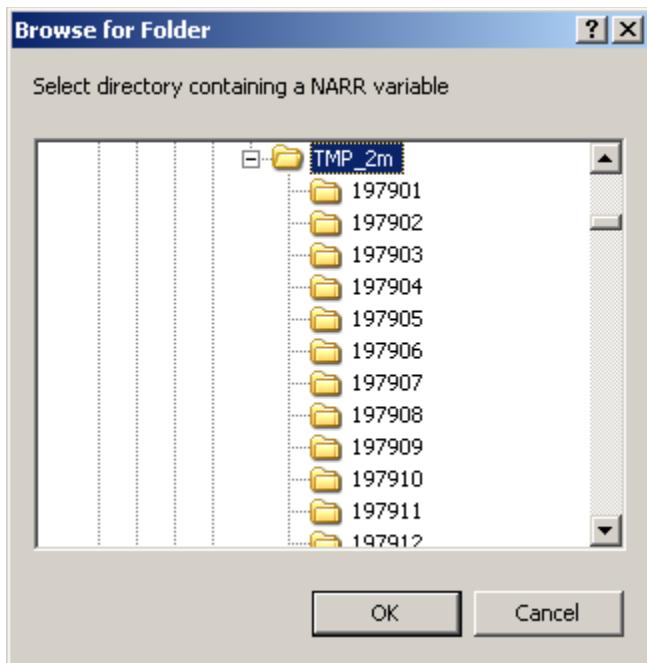


Figure 3.23: Set the directory containing the NARR variable

3. The source data parameters are displayed in the upper part of the NARR import dialog. These include the time span of the data (i.e. 1979 to 2006 inclusive), the DeltaT or timestep (i.e. 3 hours) and the indices of the source data grid (i.e. 349 by 277).

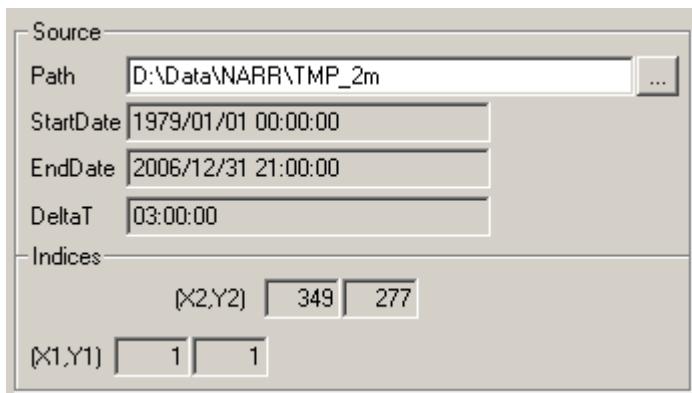


Figure 3.24: The source parameters of the NARR data

4. The NARR data can be subsetted both spatially and temporally for output. The source grid ranges from indices (1,1) at the bottom left corner to (349, 277) at the top right corner. You can subset the grid spatially or choose the full extents by editing the (X1,Y1) or (X2,Y2) indices. To subset data temporally for output, you can edit the **StartDate** or **EndDate** fields.

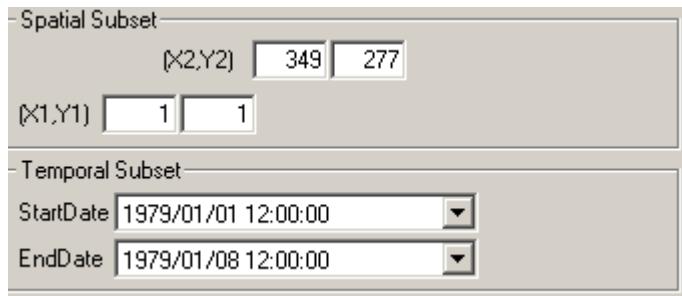


Figure 3.25: Subsetting the NARR data

5. Select the path and filename for the output grid clicking on the browse button [...] or by typing the path and filename directly into the window. You can save the output as a Rectangular grid (*.r2s) or a Rectangular Cell grid (*.r2c). Select the **Apply** button to estimate the output file size and display the output frame count.

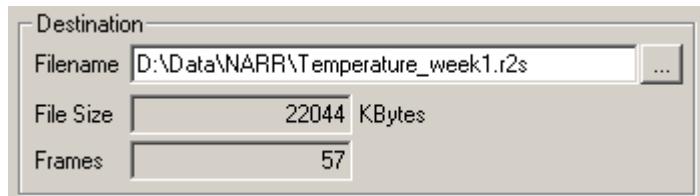


Figure 3.26: Setting the output destination for NARR data

6. Click on the **OK** button to save the output file.

4 THE GEN1D MODEL

The theoretical basis upon which GEN1D was formulated is described by Prandle and Crookshank (1972). Incorporated into the model are ideas that originated earlier with Kamphuis (1968) and Crookshank (1971).

Briefly, the one-dimensional shallow water wave equations of motion and continuity are solved by an explicit finite-difference scheme using central differences in a staggered grid system. Water elevations and velocities are solved at alternating grid points in both the spatial and temporal domains.

A multi-attribute EnSim Network object is used to carry all the information for the model. The EnSim environment allows you to describe the location and type of each model boundary as well as the location of internal channel junctions. The boundary types allowed in the model include constant or time-varying water elevation boundaries, constant or time-varying discharge boundaries, and reflective boundaries. Internal junctions can be either diverging or converging as defined by the direction of the computation. The network permits even complex river topographies to be schematized and modelled.

4.1 GENERAL BACKGROUND

4.1.1 Basic Equations

The basic equations used in GEN1D are the one-dimensional shallow water equations (Navier-Stokes equations under the hydrostatic assumption) for unsteady flow in open channels.

- The motion equation

$$\frac{\delta U}{\delta t} + U \frac{\delta U}{\delta x} + g \frac{\delta H}{\delta x} + g \frac{1}{C^2 R} U |U| = \nu \frac{\delta^2 U}{\delta x^2}$$

- The continuity equation

$$A \frac{\delta U}{\delta x} + B \frac{\delta H}{\delta t} - q_T = 0$$

where

- x = longitudinal distance along the river [m]
- t = time [s]
- H = elevation of water surface above a specific datum [m]
- U = mean cross-sectional velocity in the channel direction [m/s]
- g = acceleration due to gravity [m/s^2]
- ν = viscosity coefficient

- C = Chézy friction coefficient
- R = hydraulic radius [m]
- A = channel area [m^2]
- B = channel width [m] (values between B_{CD} and B_T)
- q_T = tributary discharge per unit length [m^2/s]

In GEN1D, the hydraulic radius, R , defined as the ratio between the channel area, A , and wetted perimeter, is approximated to the ratio between the channel area A and the channel width B .

4.1.2 Geometric Requirements

The formulation of a model based on the finite-difference method requires that discrete segments be used to represent physical topography, or geomorphology. The model consists of simulation parameters, a channels object, and the upstream and downstream boundary nodes.

The network object consists of both channel segments and nodes. The nodes are a series of points that link the segments to one another. Although the Z-component of the node is usually set to the level of the channel bottom for a node, this value is only used to assist in visualization. The model uses the cross-sections of the adjacent segments to determine the level of the channel bottom at a node. The segment is defined as the channel reach between two nodes, and is associated with a single cross-section object. The cross-section associated with a segment applies to the entire length of the segment.

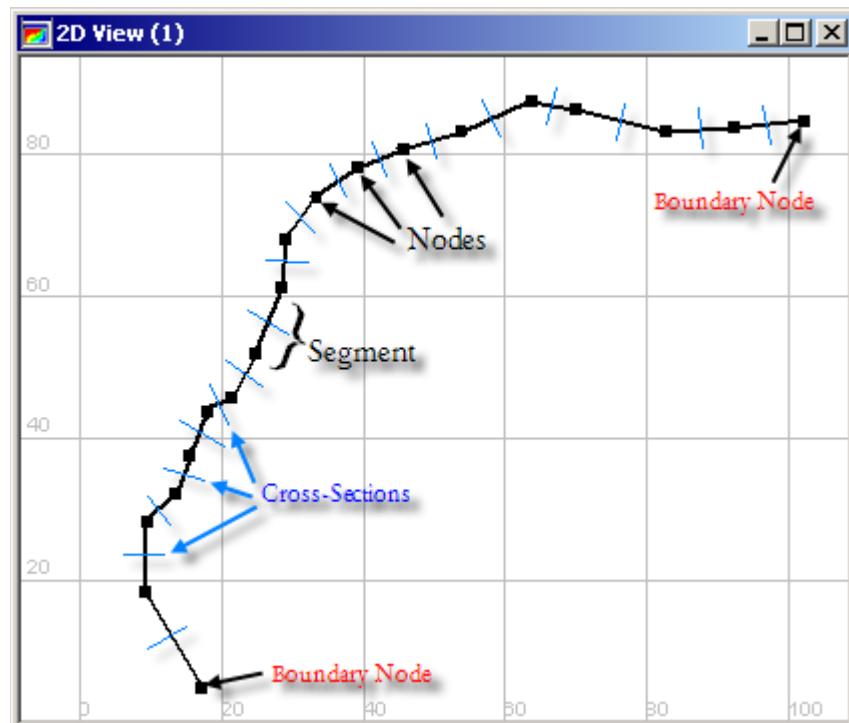


Figure 4.1: The network object is shown in black, while the cross-section object is shown in blue

For additional information on cross-sections, see "Channel" under Setting Up Simulation Parameters, on p. 223.

As shown in Figure 4.2, the cross-section is used to determine the cross-sectional parameters: cross-sectional area, channel width, and elevation. The average water elevation between end nodes of a segment is the elevation used to calculate the cross-section parameters for that segment.

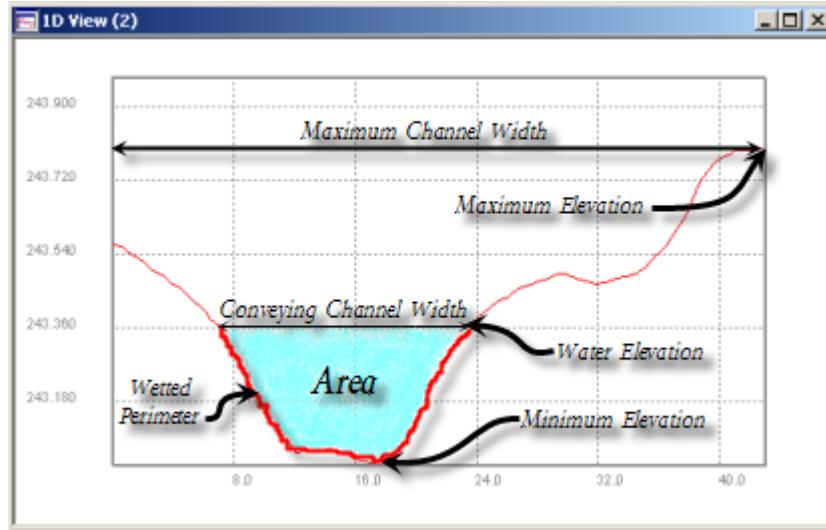


Figure 4.2: The parameters of the channel at the cross-section are applied to the entire segment

The terms used in Figure 4.2 may require some clarification:

- **Area:** The cross-sectional area of the channel, i.e., the area encompassed by the water surface and the wetted perimeter.
- **Wetted Perimeter:** The portion of the cross-section that is in contact with the water.
- **Water Elevation:** The average elevation between end nodes of the segment.
- **Maximum Elevation:** The maximum elevation of the cross-section. If the elevation exceeds the maximum elevation, the additional cross-sectional area is assumed to be rectangular. So, the additional area is equal to the maximum channel width multiplied by the height above the maximum elevation.
- **Minimum Elevation:** The minimum elevation of the cross-section. This is the point at which the cross-sectional area is equal to zero.
- **Conveying Channel Width:** The width of the channel at the current water elevation.
- **Maximum Channel Width:** The width of the channel at the maximum water elevation.

Each segment of the network is linked to two attributes:

- **Velocity:** The mean water velocity through the segment.
- **Strickler Friction:** A coefficient used to describe the level of friction that applies within a segment.

Each node of the network is linked to two additional attributes:

- **Surface Elevation:** The elevation of the water surface at the node.
- **Tributary Discharge:** Any additional influx or source of water at the node, such as a tributary entering the main channel.

The boundary nodes are defined as the end nodes of the network object. For additional information on boundaries, see the sections "Down Boundary", on p. 226, and "Up Boundary", on p. 227.

4.2 THE GEN1D INTERFACE

4.2.1 Setting Up Simulation Parameters

The parameters for running a GEN1D simulation are contained within the GEN1D Parameter Set file (*.g1d). For more information about this file, see Appendix D, on p. 315. This file is created from the GEN1DParamSet object.

To create a new simulation parameter set, select **File→New→GEN1D Run...** from the menu bar. An empty object called **NewGEN1DRun** will appear in the WorkSpace.

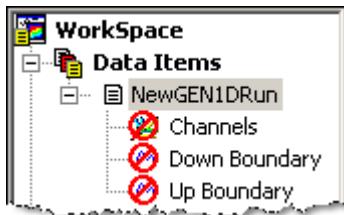


Figure 4.3: A new GEN1D Run appears in the WorkSpace with several empty components.

All data and parameters pertaining to the new simulation are specified through the Properties dialog of the **NewGEN1DRun** object.

The Properties dialog appears when the **NewGEN1DRun** object is created, or it can be opened by double-clicking on the object in the WorkSpace. The Properties dialog has four tabs:

- Simulation
- Channel
- Down Boundary
- Up Boundary

4.2.1.1 Simulation

The **Simulation** tab is used to manage parameters regarding the overall simulation.

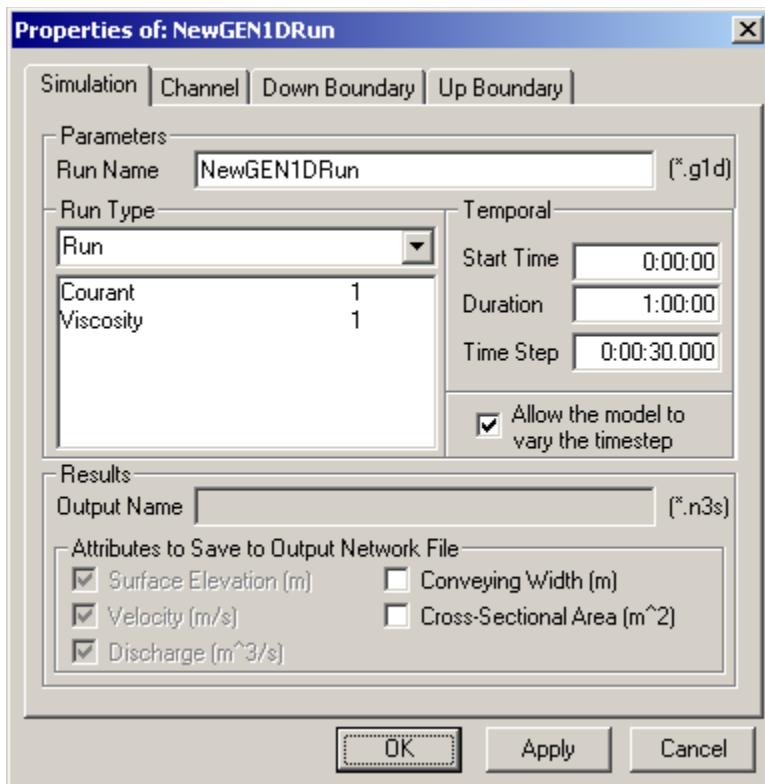


Figure 4.4: The Simulation tab controls the overall simulation run

There are two main sections to the **Simulation** tab:

The upper section, **Parameters**, controls some of the general simulation information

- **Run Name:** This is the file name of the GEN1D parameter file (*.g1d). This name is also used as the root for naming generated output files.
- **Temporal:** This area controls the time step and duration of the simulation.
 - **Start Time:** The time in the simulation at which calculations begin. The format is hours: minutes: seconds.
 - **Duration:** The length of time to be simulated. The format is hours: minutes: seconds.
 - **Time Step:** The interval between calculation of results during the simulation. The format is hours: minutes: seconds.
 - **Allow the model to vary the timestep:** If this box is checked, the simulation will vary the time step as needed to complete the simulation in a reasonable amount of time. If results are varying widely between time steps, the interval will be decreased; if they are not varying at all, it will be increased.
- **Run Type:** This selection determines the type of simulation to be performed, which in turn determines which coefficients will be involved. There are five Run Types.



Figure 4.5: The Run Type menu indicates what type of simulation is being prepared

- **Run:** A standard simulation run, which involves one or two coefficients.
- **Courant:** The Courant number is the ratio of the physical speed of the model to the calculation speed. For a GEN1D model, this value must be greater than 0 and less than or equal to 1. This parameter is only visible when the **Allow the model to vary the timestep** option has been activated.
- **Viscosity:** The viscosity coefficient from the motion equation. Water is assumed to have a viscosity value of 1. See "Basic Equations" under General Background, on p. 215, for more information.
- **Run to Steady State:** This type of simulation generates a model of the channel under steady state conditions. This model can then be used as a starting point in further simulations to establish behaviour under atypical conditions. This type of simulation involves two or three parameters.
 - **Steady State Discharge Accuracy:** This parameter determines the point at which the model is assumed to have reached a steady state. The smaller the value, the more precise the determination. The default value of 0.0001 represents a 0.01% variation. Extremely small values, such as 1e-07, are recommended.
 - **Courant:** See **Run Type: Run**, above.
 - **Viscosity:** See **Run Type: Run**, above.
- **Calibrate Friction To Water Level:** This type of simulation is used to calibrate a friction coefficient for a channel, if the surface elevation is known.
 - **Calibrate at Node ID:** This value represents the ID of the target node to which the calibration is to be applied.
 - **Calibrate to Water Level:** This value represents the target water level at the node specified above, in metres.
 - **Minimum Strickler:** This is a lower calibration limit for the estimated Strickler value of the channel.
 - **Maximum Strickler:** This is the upper calibration limit for the estimated Strickler value of the channel.
 - **Steady State Discharge Accuracy:** See **Run Type: Run to Steady State**, above.
 - **Courant:** See **Run Type: Run**, above.
 - **Viscosity:** See **Run Type: Run**, above.

- **Calibrate Friction To Discharge:** This type of simulation is used to determine the friction coefficient for a channel, if only the discharge is known.
- **Calibrate to Discharge:** This value represents the discharge rate of the channel, in m^3/s .
- **Minimum Strickler:** This is a lower calibration limit for the estimated Strickler value of the channel.
- **Maximum Strickler:** This is the upper calibration limit for the estimated Strickler value of the channel.
- **Steady State Discharge Accuracy:** See the **Run to Steady State** Run Type.
- **Courant:** See **Run Type: Run**, above.
- **Viscosity:** See **Run Type: Run**, above.
- **Generate Rating Curve:** This type of simulation generates a list of results that can be used to produce a rating curve comparing water level to discharge.
 - **Rating Curve Node ID:** This determines the node of the channel for which the table of results will be generated.
 - **Discharge Start:** This is the initial discharge value used to generate the rating curve.
 - **Discharge Delta:** This is the interval between discharge values when generating a rating curve.
 - **Number of Trials:** This represents the number of simulations that are to be run in determining the list of results. The greater this number is, the smoother the curve will be, but the longer the simulation will take.
 - **Steady State Discharge Accuracy:** See **Run Type: Run to Steady State**, above.
 - **Courant:** See **Run Type: Run**, above.
 - **Viscosity:** See **Run Type: Run**, above.

The lower section, Results, determines how output from the simulation will be stored.

- **Output Name:** This is the name that will be used for the generated output file.
- **Attributes to Save to Output Network File:** This area determines which possible output will be saved in the network file. Surface Elevation, Velocity, and Discharge will be automatically recorded, while Conveying Width and Cross-Sectional Area are optional. Click on the checkbox next to the attribute name to include it. The optional choices are not recorded, by default.

4.2.1.2 Channel

Use this tab to specify the channel network that will be used for the simulation.

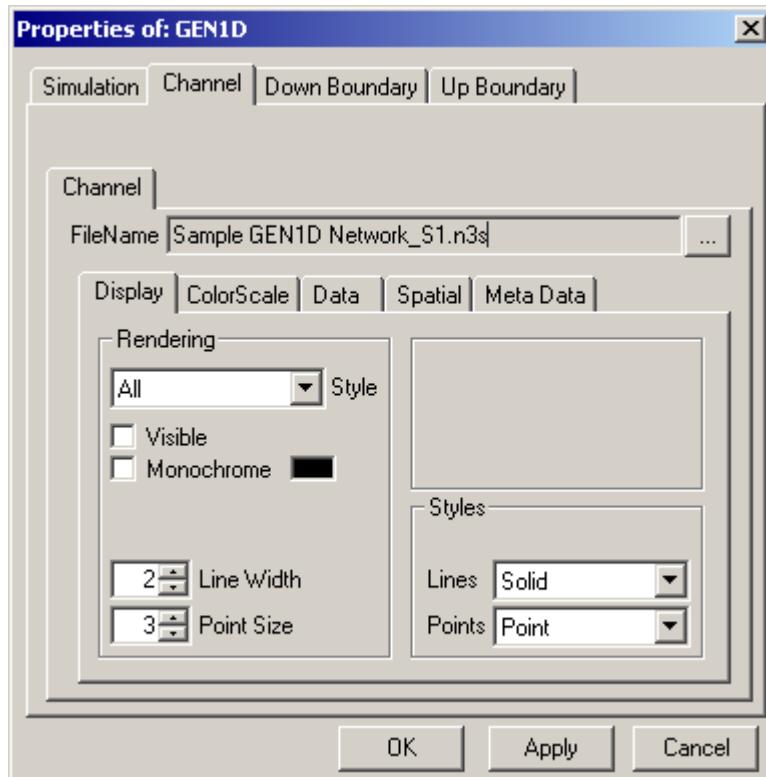


Figure 4.6: The Channel tab controls the appearance of the channel network

- **FileName:** Click the button to select the network file that describes the channel. This file must be present to run a simulation. See "Networks [n3s]", on p. 296, for more information on network (*.n3s) files. For more information on the property tabs for the channel object, see "Properties of Data Items" under Data Items, on p. 17.

4.2.1.2.1 Creating a Channel Object

To create a new channel object:

1. Click on in the **Channel** tab of the **GEN1D Run** dialog.
2. Select a 3D line set or a network file that represents the channel that is to be modelled. See "Extracting Cross-Sections from Gridded Data", on p. 123 or "Extracting Cross-Sections from Points and Line Data", on p. 124 for more information on creating a 3D Line Set.
3. If there are any attributes that are required by the model, but aren't already contained by the incoming object, they will be added at this point. The following dialog will allow you to set the default values. Attributes that are greyed out will be added, but are fixed in value.

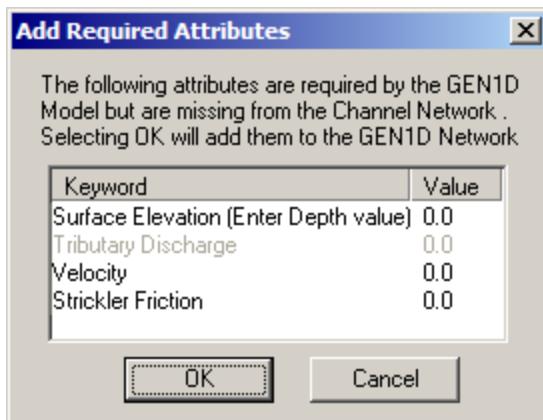


Figure 4.7: This dialog may appear if you create a channel object from a network or 3D line object

4. Next, you will be prompted to rename and resave the network object to conform to the requirements of the GEN1D model. Click **OK**.
5. Enter an appropriate name for the channel network and click **Save**.

4.2.1.2.2 Opening an Existing Channel Object

If a channel object has already been created, it can be used as the basis for the simulation.

To open an existing channel object:

1. Click on **...** in the **Channel** tab of the **GEN1D Run** dialog.
2. Select the previously created channel object. It will have the extension **.n3s**.
3. Click **OK**.

4.2.1.2.3 Changing a Segment Attribute Value

To change a segment attribute value:

1. Double-click on a segment of the network within a view to select it.
2. Right-click on the selected segment.
3. Select **Edit...** from the shortcut menu. The following dialog will appear.

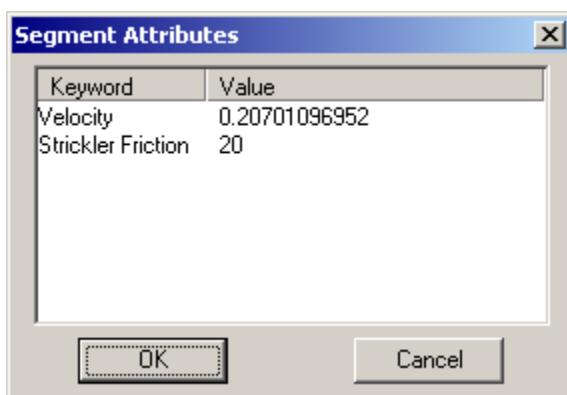


Figure 4.8: This dialog box allows you to edit a segment's attributes

4. Click on an attribute keyword or value to edit as needed.

5. Click .

4.2.1.2.4 Changing a Node Attribute Value

To change a node's attribute values:

1. Double-click on a node within a view to select it.

2. Right-click on the selected node.

3. Select **Edit...** from the shortcut menu. The following dialog will appear.

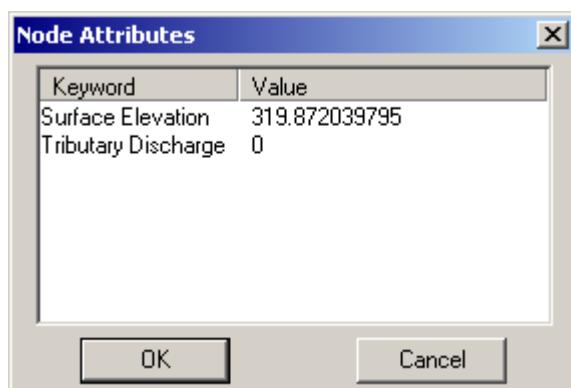


Figure 4.9: This dialog box allows you to edit the attributes of a node

4. Click on an attribute keyword or value to edit as needed.

5. Click .

To change a node's location or value:

1. Double-click on a node within a view to select it.

2. Right-click on the selected node.

3. Select **Edit Selected Point...** from the shortcut menu. The following dialog will appear.

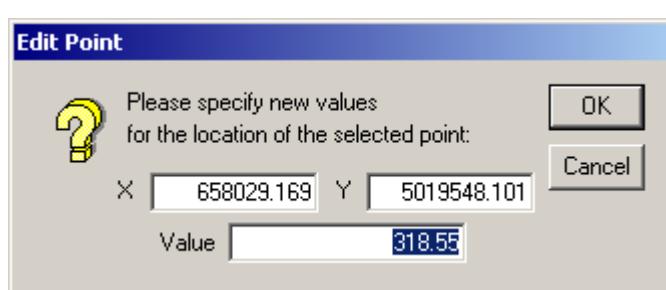


Figure 4.10: This dialog box allows you to edit location or value of a node

4. Edit the values in the dialog box.

5. Click .

4.2.1.3 Down Boundary

The **Down Boundary** tab allows you to examine and edit the properties of the downstream boundary of the model.

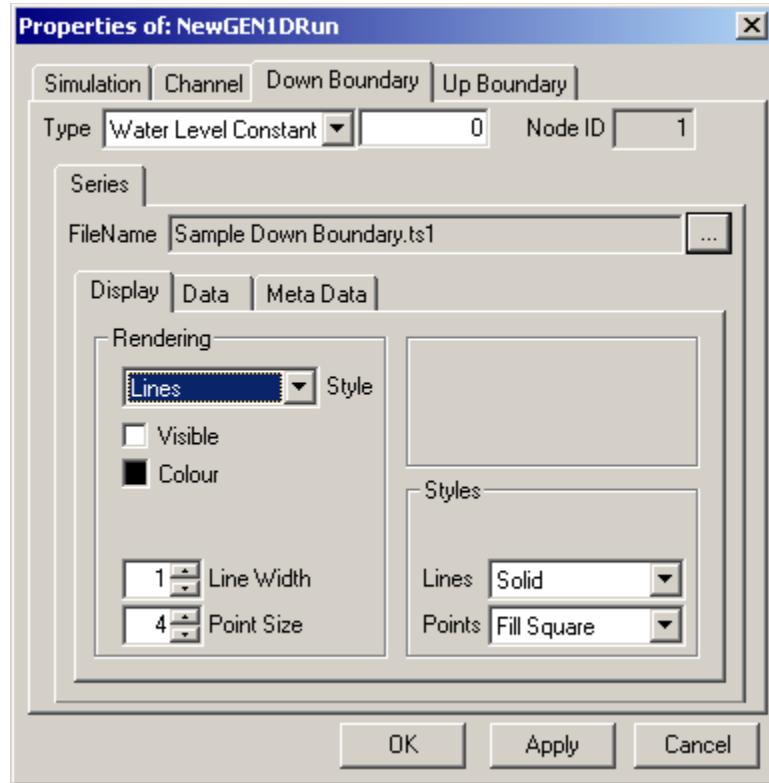


Figure 4.11: The Down Boundary tab defines the downstream boundary of the channel

There are five ways to describe the downstream boundary.

- **Type:** Use this box to select the general type of boundary at the downstream node.
 - **Water Level Constant:** If the downstream boundary is constant, enter the water level, in metres, in the text box.
 - **Water Level Series:** If the downstream boundary varies over time, click [...] to select and load a scalar time series (*.ts1) file that describes the changes in its water level.
 - **Discharge Constant:** If the downstream boundary has a constant discharge, enter the value, in m^3/s , in the text box.
 - **Discharge Series:** If the downstream boundary has a discharge that varies over time, click [...] to select and load a scalar time series (*.ts1) file that describes the changes.
 - **Free Flow:** If the downstream boundary has an effectively unlimited rate of discharge, select this option.
- **Node ID:** The downstream boundary is always considered to be Node 1 of the network.
- **Series:** Click [...] to select and load a scalar time series that describes the changes in the boundary conditions over time, if you've selected a **Water Level Series** or **Discharge Series**

boundary type. For more information on the property tabs for the time series, see "Properties of Data Items" under Data Items, on p. 17.

4.2.1.4 Up Boundary

The **Up Boundary** tab allows you to examine and edit the characteristics of the upstream boundary of the channel object.

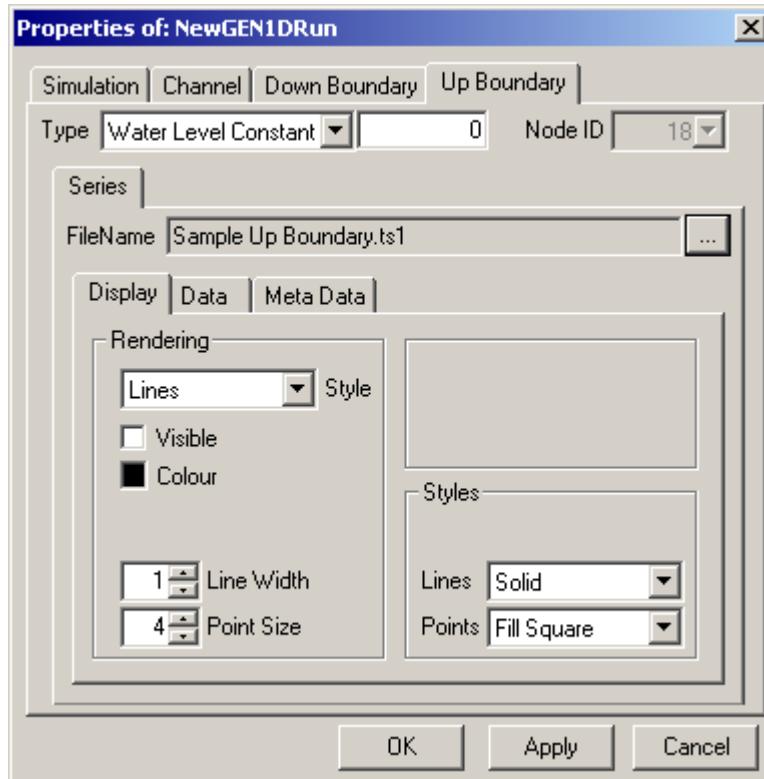


Figure 4.12: The Up Boundary tab describes the properties of the upstream boundary

Like the downstream boundary, there are five ways to describe the properties of the upstream boundary.

- **Type:** Use this box to select the general type of boundary at the upstream node of the channel.
 - **Water Level Constant:** If the upstream boundary is constant, enter the water level, in metres, in the text box.
 - **Water Level Series:** If the upstream boundary varies over time, click to select and load a scalar time series (*.ts1) file that describes the changes in its water level.
 - **Discharge Constant:** If the upstream boundary has a constant discharge, enter the value, in m³/s, in the text box.
 - **Discharge Series:** If the upstream boundary has a discharge that varies over time, click to select and load a scalar time series (*.ts1) file that describes the changes.

- **Reflective:** This boundary type indicates that there is no discharge entering the channel from this node. It effectively acts as a barrier to flow.
- **Node ID:** The upstream boundary is automatically defined as the highest node of the network, opposite the downstream boundary.
- **Series:** Click to select and load a scalar time series that describes the changes in the boundary conditions over time, if you've selected a **Water Level Series** or **Discharge Series** boundary type. For more information on the property tabs for the time series, see "Properties of Data Items" under Data Items, on p. 17.

4.2.1.5 Cross-Sections

After the simulation parameters have been established, but before the simulation can be run, cross-sectional data must be entered for each segment of the channel network. There are several ways to accomplish this.

4.2.1.5.1 Associating a Cross-Section with a Segment

If you have collected data about the bathymetry of the channel, that data can be associated with the segment to which it applies.

To associate a cross-section with a segment:

1. Load the cross-section into the WorkSpace as an i3s (3D line set) file. See "Extracting Cross-Sections from Gridded Data" under How To - Hints and Tricks, on p. 123 or "Line Sets [i2s / i3s]", on p. 279 for more information on creating a 3D line object.
2. Within the WorkSpace, drag the 3D line object containing the cross-section data onto the Channel object within the GEN1D container.
3. With the Channel object visible in a 2D or 3D view window, select the segment with which the cross-section is to be associated.
4. From the segment's shortcut menu, select **CrossSection→Associate CrossSection at Segment....**

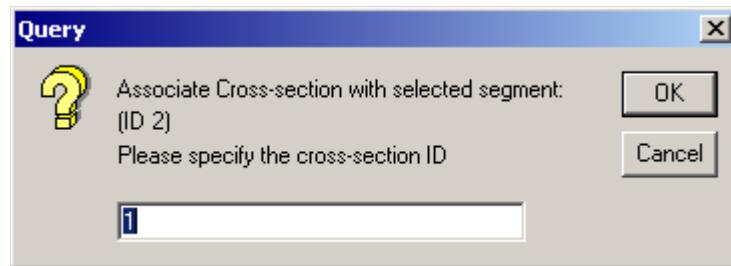


Figure 4.13: Use this dialog to select a cross-section to associate with a segment

5. In the dialog box, enter the ID of the cross-section to be associated with the segment. Lines within the file are numbered in the order in which they were created.
6. Click .

4.2.1.5.2 Scaling a Cross-Section

Scaling a cross-section allows you to multiply the entire cross-section by a single factor.

To scale a cross-section:

1. With the Channel object visible in a 2D or 3D view, select the cross-section to which you would like to apply the scaling factor.
2. From the cross-section's shortcut menu, select **Scale....**

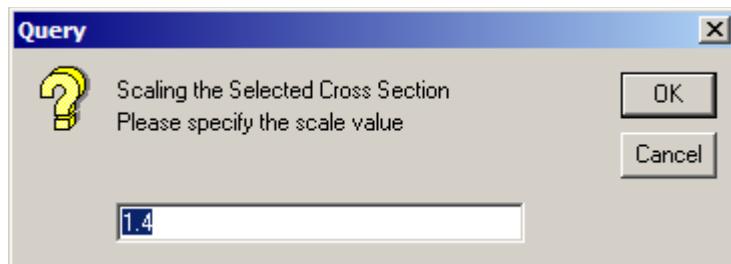


Figure 4.14: This dialog allows you to enter the scaling factor

3. Enter the multiplier you would like to apply to the cross-section and click **OK**.

4.2.1.5.3 Copying a Cross-section to a Segment

If multiple segments have similar cross-section data, it is possible to copy the cross-sections directly from one segment to another.

To copy a cross-section to a segment:

1. Select the segment to which you would like to copy the cross-section.

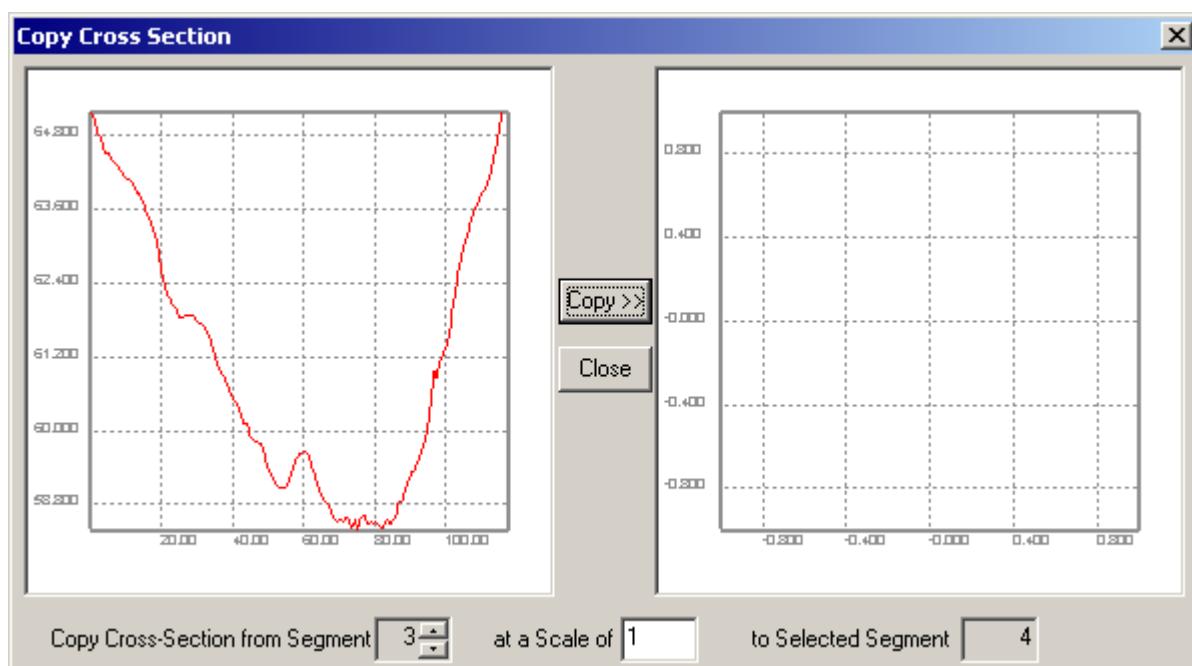


Figure 4.15: This dialog is used to copy cross-section data directly from one segment of a channel to another

2. From the shortcut menu, select **CrossSection**→**Copy CrossSection to Segment...**

3. Select a scale, if the segments are of different sizes, and click **Copy >>**.

4.2.1.5.4 Orthogonally Positioning a Cross-Section

If a created, copied, or otherwise associated cross-section is not properly positioned with regards to its segment, it can be automatically moved horizontally (on the x-y plane) so that the cross-section meets the midpoint of the segment at a right angle.

Note: The (x, y) position of the lowest point of the cross-section is set to the midpoint of the segment.

To orthogonally position a cross-section:

1. Within a 2D or 3D view window, select either the segment or the cross-section that is not properly positioned.
2. From the shortcut menu, select **CrossSection**→**Orthogonally Position CrossSection at Segment**, if you selected the segment, or **Orthogonally Position**, if you selected the cross-section.

4.2.1.5.5 Resampling a Cross-Section

If the data used to define a cross-section is too detailed, or not detailed enough, the line can be resampled, producing a cross-section containing more or fewer points.

To resample a cross-section:

1. Within the View window, select the cross-section that you'd like to resample.
2. From the shortcut menu, select **Resample**, followed by **Max-Distance...** (to increase the number of points on the line) or **Equi-Distance...** (to decrease the number of points).

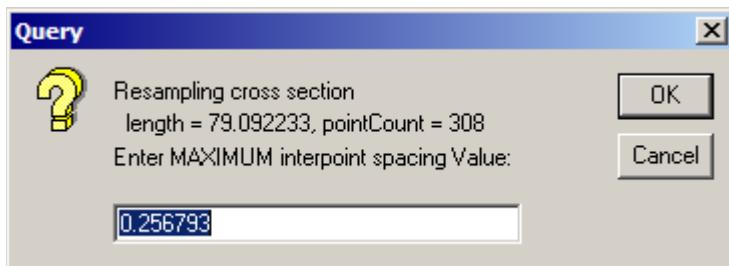


Figure 4.16: This dialog allows you to enter a Maximum interpoint spacing value

- For a **Max-Distance** resampling, the number to be entered represents the maximum distance that will be allowed between any two points on the line. If any points are further apart than this, additional points will be inserted to decrease the distance between them.

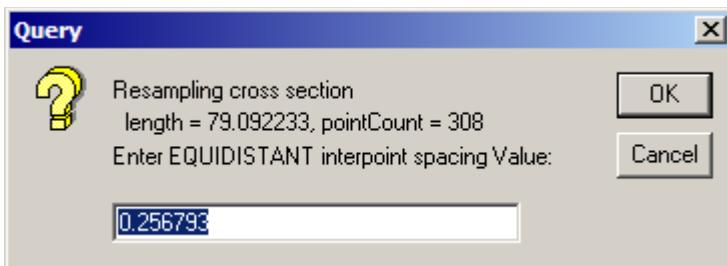


Figure 4.17: This dialog allows you to enter an Equidistant interpoint spacing value

- For an **Equi-Distance** resampling, the number to be entered represents the distance between each point. If you enter a number greater than the existing value, the cross-section will contain fewer points, while a smaller number will produce a cross-section containing more points. The length is divided by the value entered to determine the new pointCount value, and the points are equally spaced along the line.

3. After entering the new value, click **OK**.

4.2.1.5.6 Interpolating a Cross-Section

If data is available for the segments on either side of a cross-section, that data can be used to produce a cross-section for a segment that resembles a combination of its neighbours'.

To interpolate data from two cross-sections:

1. With the Channel object visible in a 2D or 3D view, select the segment to which you would like to assign the combined cross-section. The segment must have neighbours on both sides.
2. From the segment's shortcut menu, select **CrossSection→Interpolate CrossSection at Segment...**

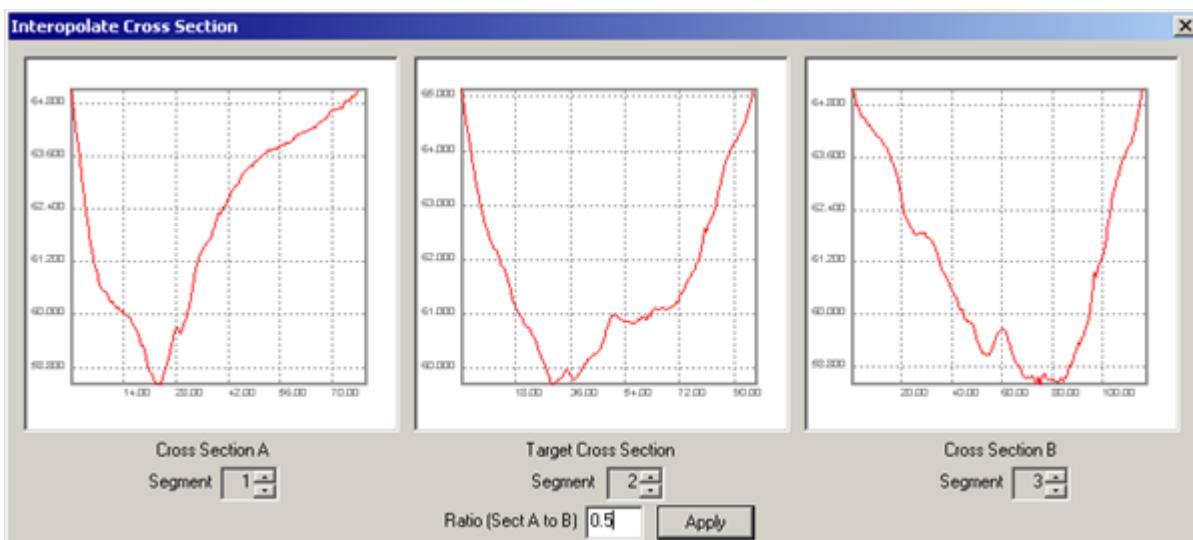


Figure 4.18: The centre cross-section is a combination of the left and right cross-sections

3. Use the arrows to select two source cross-sections from the channel.

4. Enter a value in the dialog that reflects the degree to which the created cross-section should resemble each of the sources. A value of 0.5 produces an equal combination of the two sources. Lower values produce a result that more closely resembles cross-section A, while higher values produce a result that resembles cross-section B. The value must be between 0 (identical to cross-section A) and 1 (identical to cross-section B).

5. Click **Apply**.

4.2.1.5.7 Vertically Offsetting a Cross-Section

Vertically offsetting a cross-section can be used to adjust the elevation of an entire cross-section at the same time. When you enter the vertical offset value, every point on the line has its elevation increased or decreased appropriately.

To vertically offset a cross-section:

1. Within a View window, select the cross-section you'd like to offset.
2. From the shortcut menu, select **Vertical Offset...**
3. Enter the value, in metres, by which the cross-section should be elevated. To lower the cross-section (i.e., to reduce its elevation), enter a negative number.
4. Click **OK**.

4.2.1.5.8 Generating a Simple Cross-Section

If no data exists for a given segment's cross-section, and no other option is available, a rough approximation of the possible cross-section can be produced.

To generate a simple cross-section:

1. With the Channel object visible in a 2D or 3D view, select the segment for which you would like to generate a simple cross-section.
2. From the segment's shortcut menu, select **CrossSection→Generate CrossSection at Segment....**

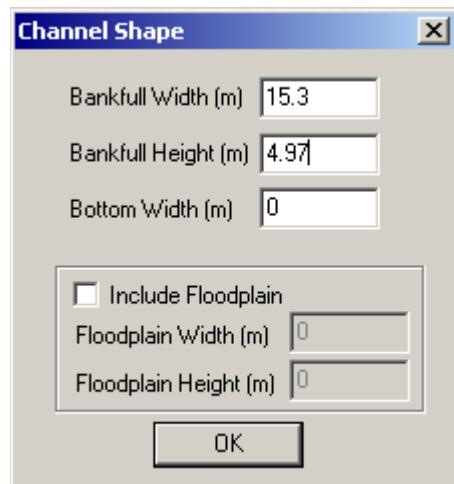


Figure 4.19: This dialog allows you to create a simple cross-section

3. Enter the estimated values for the cross-section of the segment.

4. Click .

4.2.1.5.9 Removing a Cross-Section

This option is available in case you want to remove a cross-section from the Channels object.

To remove a cross-section:

1. Within a View window, select the cross-section that you'd like to remove.
2. From the cross-section's shortcut menu, select **Remove**.
3. You will be asked to confirm your selection. Remember, once you've removed a cross-section, it cannot be recovered. Click  to remove the cross-section, or  to abort the removal.

4.2.1.5.10 Cross-Section Properties

The Cross-Section properties window can be used to examine the profile of a cross-section, and to see what the effects of varying water levels are on its parameters.

To view the properties of a cross-section:

1. Within a View window, select the cross-section whose properties you'd like to examine.
2. From the shortcut menu, select **Properties....**

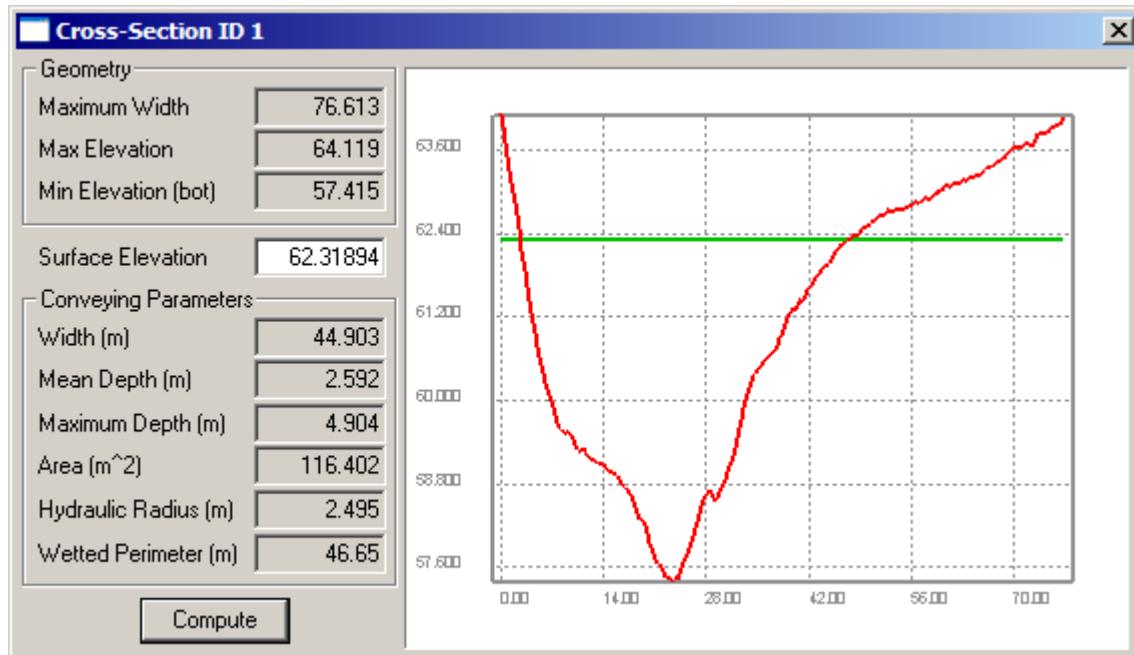


Figure 4.20: The Cross-Section Properties window shows the numerical characteristics of the cross-section

Note: You can recalculate the values shown by entering a new value for the Surface Elevation and clicking the .

3. Click the  button to close the window.

4.2.2 Running the GEN1D Model

To run a GEN1D simulation:

1. Make sure that all mandatory fields on the **Simulation**, **Channel**, **Down Boundary**, and **Up Boundary** tabs are complete.
2. Select the GEN1D Parameter file object in the WorkSpace.
3. Select **Run→Check Parameters** from the menu bar.
 - If any information is missing, a dialog box will appear, listing the missing parameters.

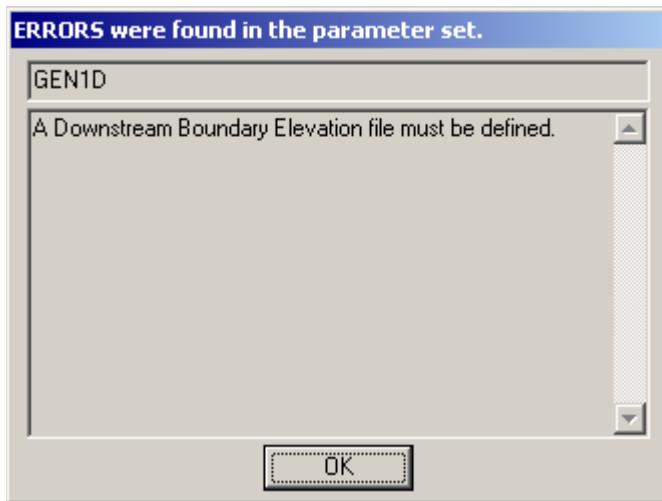


Figure 4.21: This dialog box lists any missing parameters

- If there are no missing parameters, a confirmation dialog will appear.

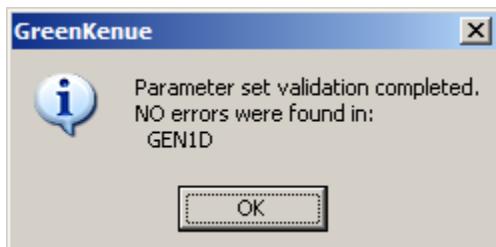


Figure 4.22: This dialog box appears if all parameters have been entered

4. Select **Run→Launch Simulation** from the menu bar.

4.2.3 Displaying Simulation Output

GEN1D produces a single output file, regardless of the type of simulation run. The output file produced is a time-varying network file, which is displayed in the WorkSpace as a child of the parameter object. The Out Network object has the same geometry as the channel object that was used as input to the simulation.

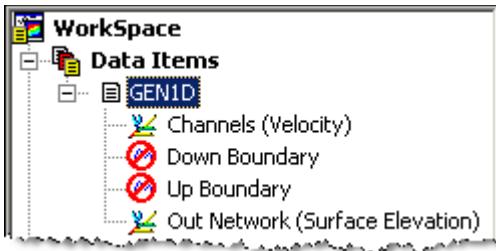


Figure 4.23: The Out Network object is the output of a simulation

If the simulation run was a normal **Run**, a **Run To Steady State**, or a **Generate Rating Curve** Run Type, the Out Network object will have three data attributes:

- **Velocity:** This is the velocity of water within a channel segment, in m/s.
- **Discharge:** This is the total discharge from a channel segment, in m³/s.
- **Surface Elevation:** This is the elevation of the surface of the water in a channel segment, in metres.

If the simulation was a **Calibrate Friction to Water Level** or a **Calibrate Friction to Discharge** Run Type, then the Out Network object will have a fourth data attribute:

- **Strickler Friction:** This is the calculated Strickler Friction coefficient for a channel segment.

If a simulation has already been run for a particular GEN1D Parameter object, you can load the results by selecting **Run→Load Results** from the menu bar.

The Out Network object can be displayed in a 2D or 3D view, and can be animated. The number of frames will depend on the parameters chosen for the simulation. For more information on viewing time-varying data, see "Animation" under Views, on p. 61.

4.2.3.1 Creating a Hot Start From an Output

In some cases, it may be useful to use the results of a simulation as input for another simulation. This is particularly common with **Run To Steady State** simulations.

To extract a Hot Start from a GEN1D model run:

1. After a simulation has completed, right-click on the GEN1D Parameter object in the WorkSpace.
2. From the shortcut menu, select **Create Hot Start Network...** A dialog will appear.
3. Enter the number of the frame that you would like to use as input to another simulation.

Note: The first frame is numbered 0.

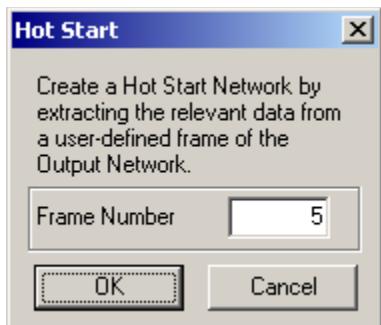


Figure 4.24: Use this dialog to create a Hot Start network

4. Click **OK**.
5. Save the Out Network object by selecting **File→Save** from the menu bar, or by clicking .

When you create a new GEN1D Parameter object, use the Hot Start network file as the Channel object. This file contains the velocities, discharge, and surface elevation at the chosen frame. These conditions are then used as the initial conditions for the new GEN1D model.

5 THE HBV-EC MODEL

The HBV model is a conceptual watershed model developed by SMHI, the Swedish Meteorological and Hydrological Institute. The HBV-EC model was adapted by Environment Canada and UBC to simulate watershed response in mountainous terrain, as well as other environments.

Green Kenu provides the model with a two-dimensional interface, allowing you to relate the output of the model back to the maps from which it was derived.

5.1 GENERAL BACKGROUND

5.1.1 Background and History of the Model

HBV is a conceptual hydrological model designed for use in mountainous environments. It was originally developed by Sten Bergström in the early 1970s while working at the Swedish Meteorological and Hydrological Institute (SMHI). Over the past 35 years, the model has been used extensively for hydrological forecasting, engineering design, and climate change studies. Bergström (1995) provides a complete description of the history and application of the model as well as details on the basic internal routines.

A comprehensive evaluation of current and proposed new routines within the HBV hydrological model was carried out by SMHI in the mid-1990s (Lindström et al., 1997). One of the primary tenets of developers of the HBV model is to add complexity to the model only when it shows an improvement in the simulation of hydrological process. The review of the HBV model by Lindström et al. (1997) updated routines for watershed runoff. However, several proposed new algorithms proved to provide little or no significant improvement in model performance.

A Canadian version of the HBV model has been maintained by Dr. Dan Moore (UBC) since the mid-1980s. Moore (1993) developed and tested a glacier routine for the model. In 2000, Dr. Moore provided the source code for the model (then written in Turbo Pascal) to Environment Canada and current model development has since been co-managed by Environment Canada and UBC. The Canadian version is referred to as the HBV-EC ("Environment Canada") model.

5.1.2 Algorithms Specific to the Model

The HBV model was modified to match changes specified in the Lindström et al. (1997) paper. The HBV-EC model represents the Canadian version of the HBV model. There are slight differences between the Swedish and Canadian versions of the model which are discussed below. The reader is directed to Bergstrom (1995), Lindstrom et al. (1997), and Moore (1993) for descriptions of model routines. This section describes the main components of HBV-EC that are distinct from other available versions of the HBV model.

5.1.2.1 Climate Zones

Climate zone representation was added to the HBV-EC model to better represent the lateral climatic gradients which may occur across a basin. Each climate zone is associated with a single climate station and unique parameter values for specifying the distribution of climate within the zone, such as temperature and precipitation lapse rates. Runoff from a climate zone is lumped through a series of fast and slow response reservoirs of similar configuration to those in the traditional HBV model (Lindström et al., 1997).

5.1.2.2 Snow Melt Factor Variation with Terrain Aspect and Slope

Within the traditional HBV model, the snow melt factor does not vary with respect to terrain slope or aspect. In the HBV-EC model, the snow melt factor varies as a function of aspect (b) and slope (s) as:

$$MF = MF_{FLAT} \times [1 - AM \cdot \sin(s) \cdot \cos(b)] \quad (1)$$

where MF_{FLAT} is the melt factor computed for flat terrain (mm/d), and AM is a model parameter representing the aspect-slope reduction factor (dimensionless) which varies between $0 \leq AM < 1$

5.1.2.3 Watershed Routing

The traditional HBV model uses a MAXBAS parameter to transform fast and slow reservoir releases into streamflow. The HBV-EC model does not use a weighting function to distribute reservoir releases from the model. Output from each reservoir is totalled to predict the discharge for the given model time step.

5.1.3 References

Bergström, S., 1995. The HBV Model. In: V.P. Singh (editor) Computer Models of Watershed Hydrology. Water Resources Publications, Highlands Ranch, Colorado pp. 443- 470.

Lindström, G, Johansson, B, Persson, M, Gardelin, M, and Bergström, S., 1997. Development and test of the distributed HBV-96 hydrological model, *J. Hydrol.*, 201, 272–288.

Moore, R.D., 1993. Application of a conceptual streamflow model in a glacierized drainage basin, *J. Hydrol.*, 150: 151-168.

5.2 THE HBV-EC INTERFACE

The parameters required to execute the HBV-EC model are stored in a data file with the extension *.HBV. For more information on the structure of this file, see "The HBV-EC Parameter Set File", on p. 323. Within the WorkSpace, the parameters are displayed as an HBV-EC Parameter Set object.

To create a new HBV-EC parameter object, select **File**→**New**→**HBV-EC Run** from the menu bar. A **New HBV-EC Parameter Set** object will appear in the WorkSpace, and the HBV-EC dialog box will open.

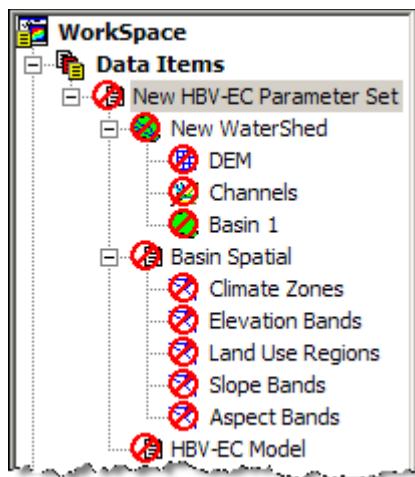


Figure 5.1: The new HBV-EC Parameter Set object contains several children objects

This **HBV-EC Parameter Set** dialog consists of three or more panels:

- **EnSim WaterShed** - This panel contains information on the watershed that is being modelled.
- **Basin** - This panel contains spatial information derived from the watershed selected on the **EnSim WaterShed** panel.
- **Simulation** - This panel contains the parameters for the model itself.
- **Climate Zone** - This panel displays (and allows you to edit) variables that describe the conditions in a particular climate zone. This panel is only displayed after the HBV-EC Model has been configured, and additional panels are displayed if more than one climate zone is used within the simulation. See "The Climate Tab", on p. 243 for more information on Climate Zones.

5.2.1 The EnSim WaterShed Panel

The **WaterShed** panel allows you to select and edit the watershed object that is to be modelled.

For detailed information on creating a watershed object, see "Creating a New Watershed Object", on p. 136.

There are two ways to prepare a watershed for the HBV-EC model:

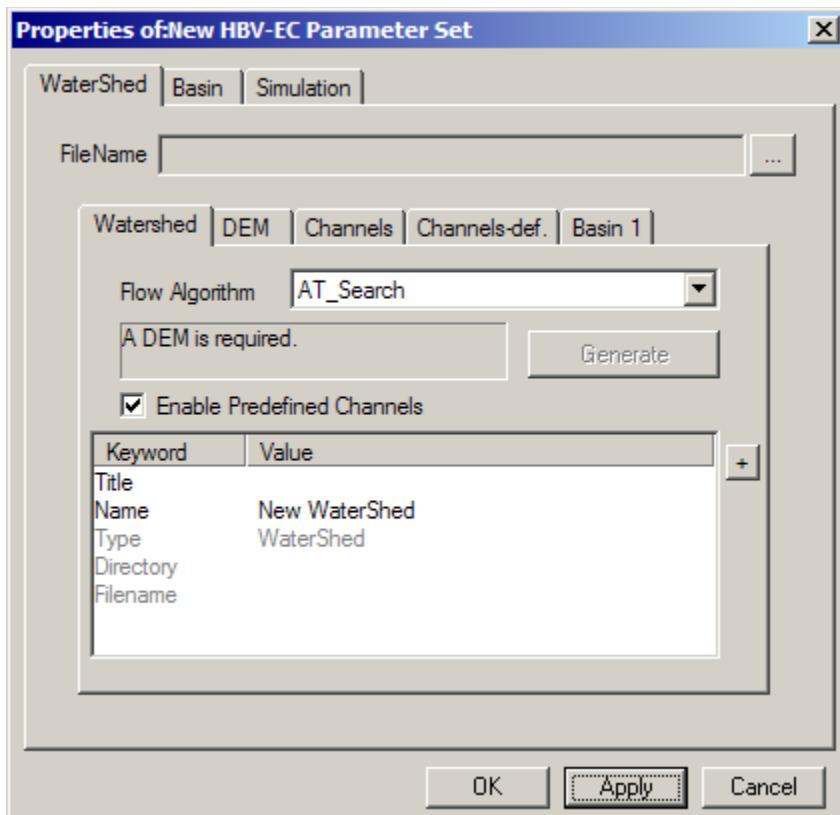


Figure 5.2: This is the WaterShed panel with the Predefined Channels option enabled

- If you've already created a watershed using Green Kenu, you can open that object and drag it to the **HBV-EC Parameter Set** object. The information on the existing watershed will be copied into the HBV-EC watershed child object, including the Channels and Basin objects. You can also select the watershed by clicking and selecting the file.

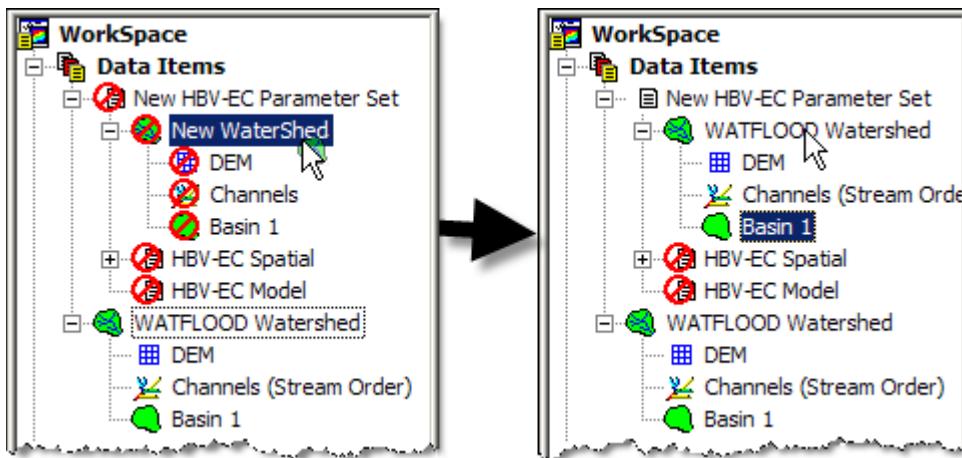


Figure 5.3: By dragging the Watershed object into the parameter set, you can copy its data

- If you don't have a pregenerated watershed object, you can load a DEM and use it to create a watershed by dragging it into the **DEM** child object of the **New Watershed** container within the **HBV-EC Parameter Set** object.

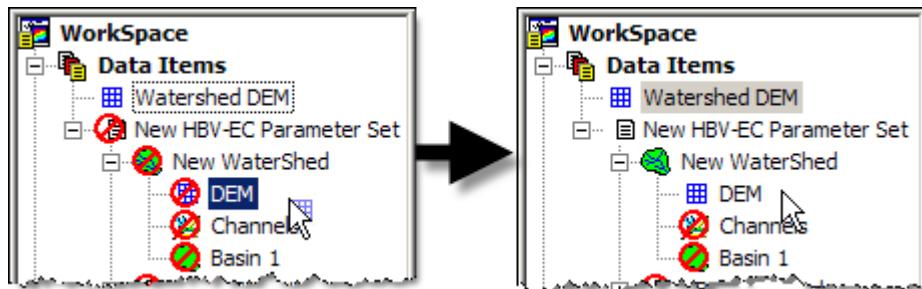


Figure 5.4: A DEM can also be added to the HBV-EC Parameter Set object by itself

Once the DEM has been added, you can use it to create the Channels and Basin objects by clicking the **Generate** button.

When you generate the Basin and Channels for a new watershed, the outlet node that is used represents the outlet of the entire DEM. If the watershed that you're modelling is contained in a smaller portion of the map, you'll have to designate the correct outlet node manually.

To identify an alternate basin object:

1. In a view window, locate the node of the **Channels** object that you would like to designate as the outlet node for the new watershed basin. Select the node by double-clicking it. The node will be identified by a pink dot.
2. Right-click the node to show its shortcut menu and select **Add Basin**. A new Basin object will be added to the WaterShed object.

Within the **WaterShed** panel, there are four (or more) tabs:

- **Watershed** - This tab lets you select the flow algorithm to be used to generate the Channels and Basin objects, enable or disable the Predefined Channels tab, and change the watershed's metadata. If you've imported an existing WATFLOOD watershed, this tab will be disabled, except for the metadata area.
- **DEM** - This tab contains the standard EnSim tabs for a rectangular grid.
- **Channels** - This tab allows you to select the criteria used to identify the channel locations on the DEM. It also contains the standard EnSim data item tabs.
- **Basin** - There is one Basin tab for each basin contained within the watershed object, named **Basin 1**, **Basin 2**, and so on. This tab allows you to control the appearance of the basin object, by means of the standard EnSim data item tabs. The extent of the basin is determined by the algorithm selected on the **Watershed** tab.

If the **Enable Predefined Channels** option has been selected on the **Watershed** tab, a fifth tab becomes available on the WaterShed panel:

- **Channels-def.** - This tab allows you to enter information on pre-existing channels within the watershed. See "Using Predefined Channels" under Watershed Objects, on p. 145 for details on using this option.

5.2.2 The Basin Panel

The **Basin** panel allows you to identify regions within the watershed. These regions include the climate zones, elevation bands, land use classes, and slope and aspect areas. The selections made on this panel will determine the number of classes to be simulated by the HBV-EC model.

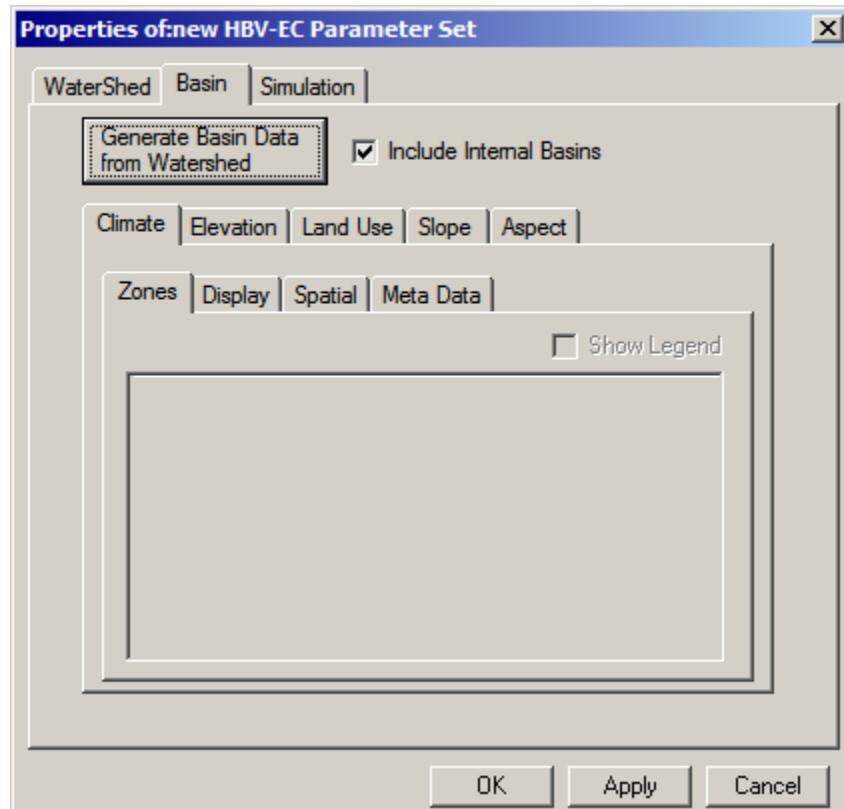


Figure 5.5: This HBV-EC panel has not yet been activated

Initially, this panel is inactive. To activate the **Basin** panel, click the **Generate Basin Data from Watershed** button. This will generate initial values for the spatial data corresponding to the watershed object.

Note: If there is more than one basin object contained in the watershed, you must select the basin that is to be used in the WorkSpace panel.

To exclude basins that are internal to the watershed, that is, to generate data for **only** the selected basin, clear the **Include Internal Basins** checkbox before clicking the **Generate Basin data from Watershed** button. If there is only one basin object in the watershed, this option is disregarded.

The **Basin** panel contains five tabs, each of which corresponds to a 2D Triangular Scalar Mesh object shown as a child of the basin object within the **HBV-EC Parameter Set** object. The triangular mesh objects can be displayed in a 3D view window.

5.2.2.1 The Climate Tab

The **Climate** tab indicates how the basin is divided into climate zones. Smaller watersheds will usually be contained entirely within a single zone, but larger areas may require multiple sets of climate data.

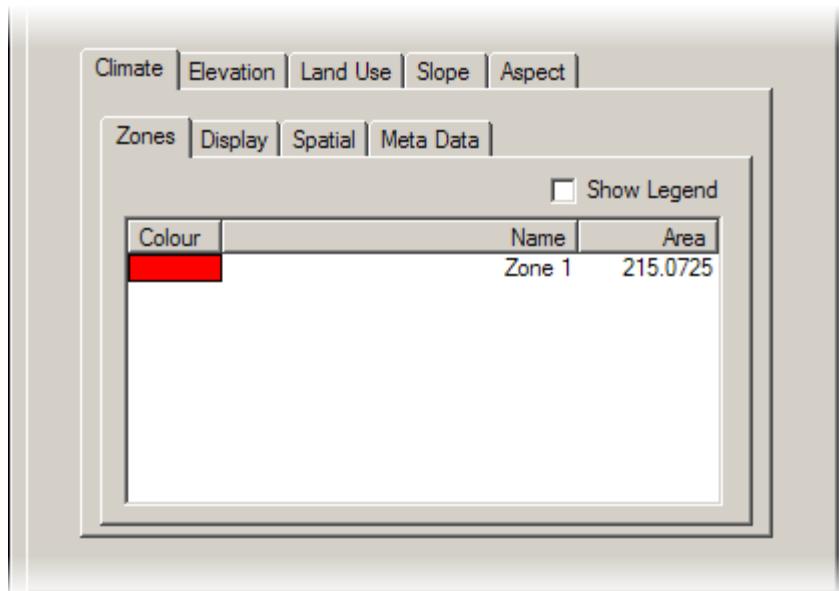


Figure 5.6: This watershed is contained within a single climate zone

To assign areas of a watershed to a different climate zone, see "Identifying Zones Within HBV-EC", on p. 248.

5.2.2.2 The Elevation Tab

The **Elevation** tab indicates which elevation bands appear in the watershed. Within the tab, you can control how many bands are used, as well as the boundaries for each band. Clicking within the **Colour** column allows you to change the colours assigned to each band.

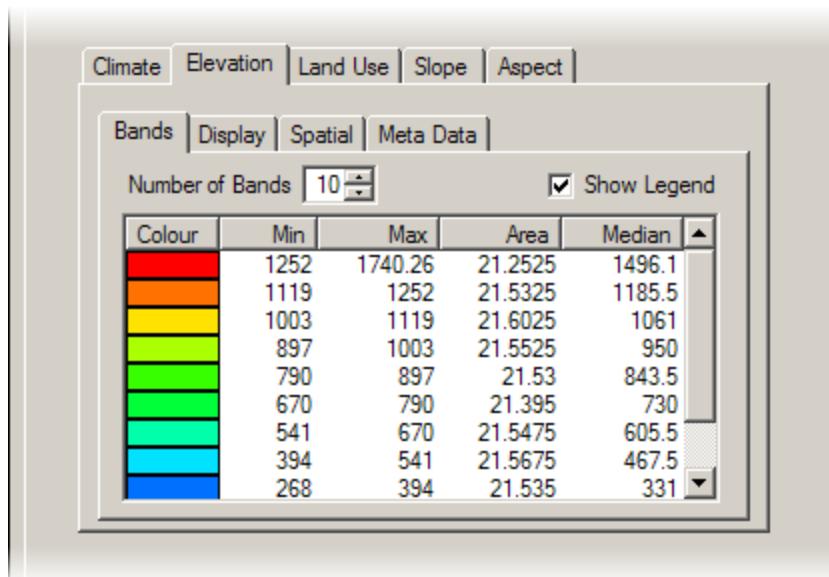


Figure 5.7: This watershed has been divided into 10 elevation bands of roughly equal area

By clicking on and changing the values in the **Min** column, you can redefine the limits of each band; the **Max**, **Area**, and **Median** values will automatically change to match. In the HBV-EC model, each elevation band is identified by a single elevation value. By default, this is the median elevation value of the zone.

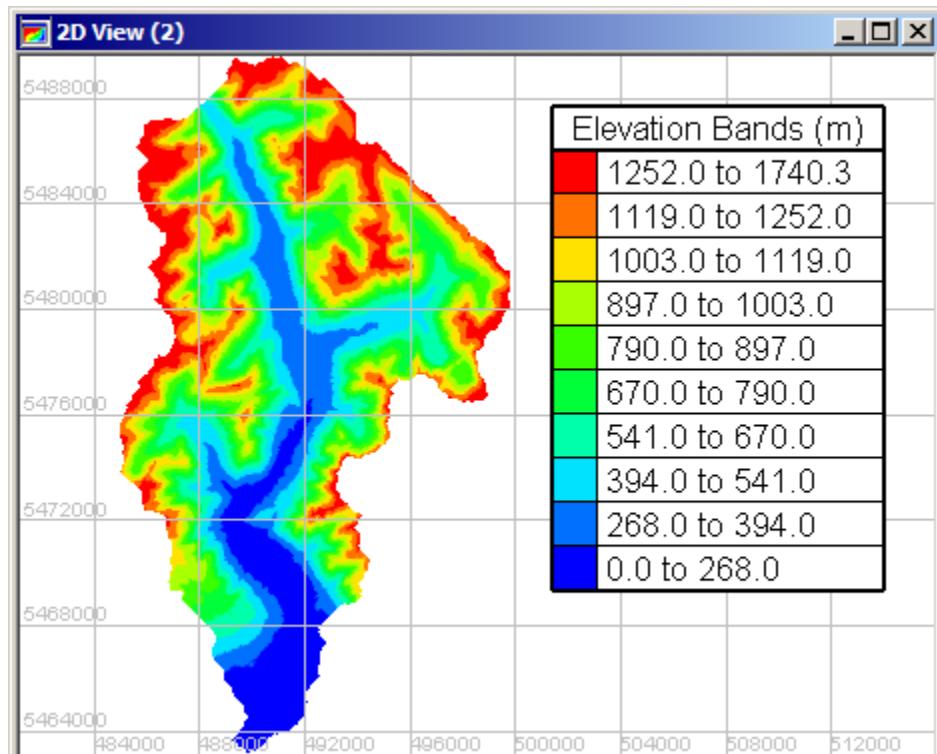


Figure 5.8: This 2D view shows the triangular mesh associated with the Elevation tab shown above

By default, a newly created **HBV-EC Parameter Set** will contain five elevation bands, each of equal elevation range.

5.2.2.3 The Land Use Tab

The **Land Use** tab allows you to examine the relative areas of each land use category. There are four categories that are of significance to HBV-EC:

- **Lake** - This category is used for water other than that located within a channel. Note: Lakes are always considered to have a slope of 0 (zero). For classification purposes, the Lake category is assigned a value of **3**.

Note: Lakes are always considered to have a slope of 0 (zero) and to fall within a single elevation band.

- **Glacier** - This category is used for terrain that is covered by snow or ice year-round. The Glacier category is assigned a value of **2**.
- **Forest** - This category is used for terrain that is covered with trees, including deciduous, coniferous, and mixed forest. The Forest category is assigned a value of **1**.
- **Open** - This category is used for terrain that has relatively little tree cover, as compared to the Forest category. This may include plains, rocky areas, or desert, among others. The Open category is assigned a value of **0**.

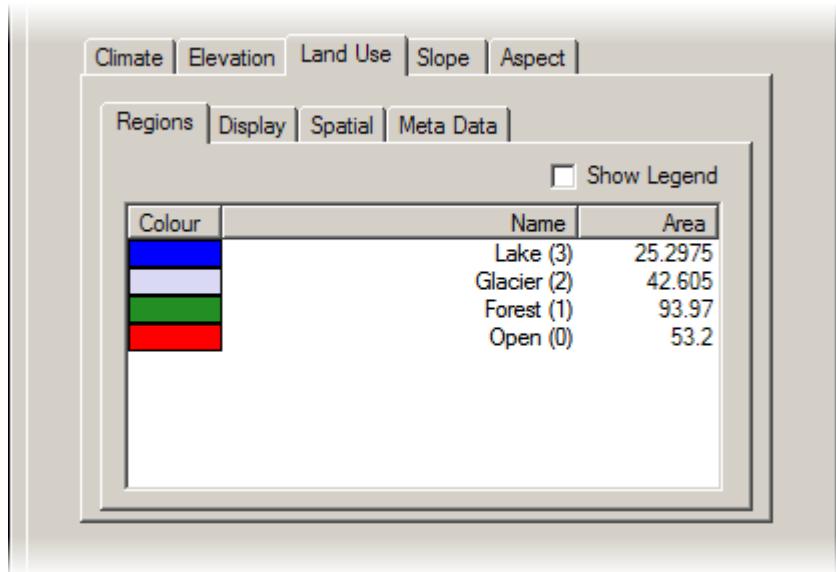


Figure 5.9: This watershed contains regions of all four terrain types, but is largely forested

To assign areas of the watershed to a different category, see "Identifying Zones Within HBV-EC", on p. 248. By default, all areas of a newly created **HBV-EC Parameter Set** watershed are assigned to the Open region.

5.2.2.4 The Slope Tab

The **Slope** tab lets you control the number of slope bands that will be used in the model, as well as the criteria for each band. Like the **Elevation** tab, you can change the **Min** values by clicking on and editing the values; the **Max**, **Area**, and **Median** values will change accordingly. You can also change the colour values by clicking within the **Colour** column.

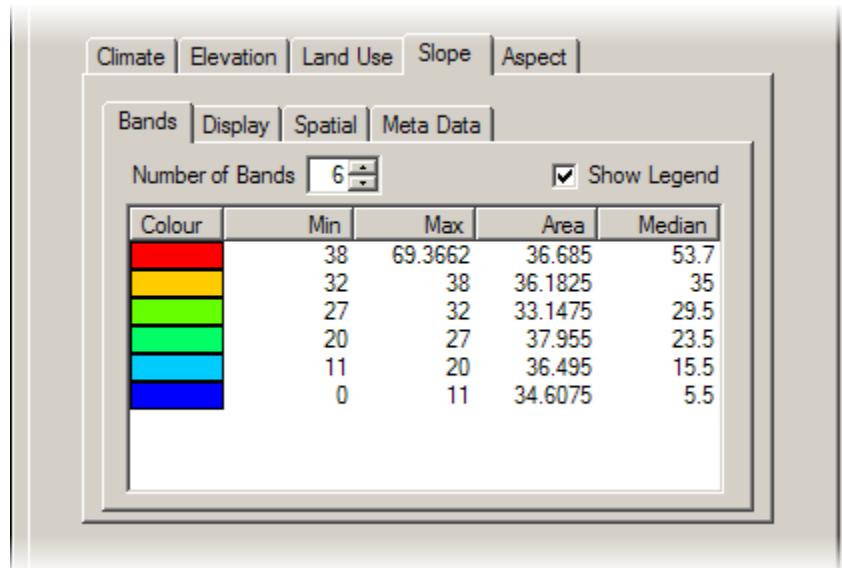


Figure 5.10: This watershed has been divided into six slope bands of roughly equal area

The numbers given for the **Min**, **Max**, and **Median** values refer to the percentage of slope, ranging from **0** (flat) to **100** (vertical). Each slope category is referred to within HBV-EC by its median value.

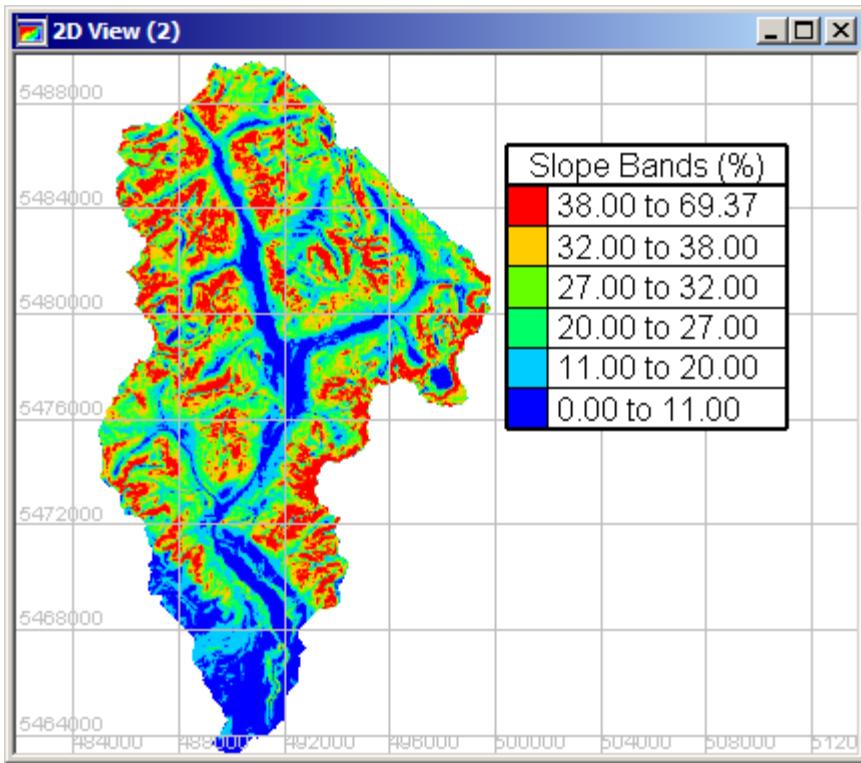


Figure 5.11: This triangular mesh corresponds to the Slope tab shown above

5.2.2.5 The Aspect Tab

The Aspect tab shows the aspect, or the direction that the slope faces, of the terrain within the watershed. This value, combined with the elevation and slope values, gives the position and orientation of each area of land within the watershed.

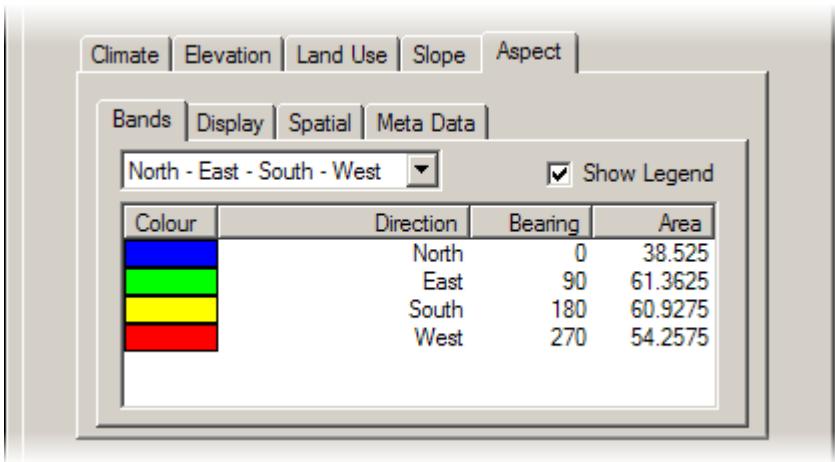


Figure 5.12: This Aspect tab divides the terrain into four aspect bands.

There are three options for the aspect category, which can be selected on the menu at the top of the tab:

- **None** - The aspect of the terrain is ignored for the simulation
- **North - South** - The terrain is divided into north-facing and south-facing categories
- **North - East - South - West** - The terrain is divided into north-facing, south-facing, east-facing, and west-facing categories.

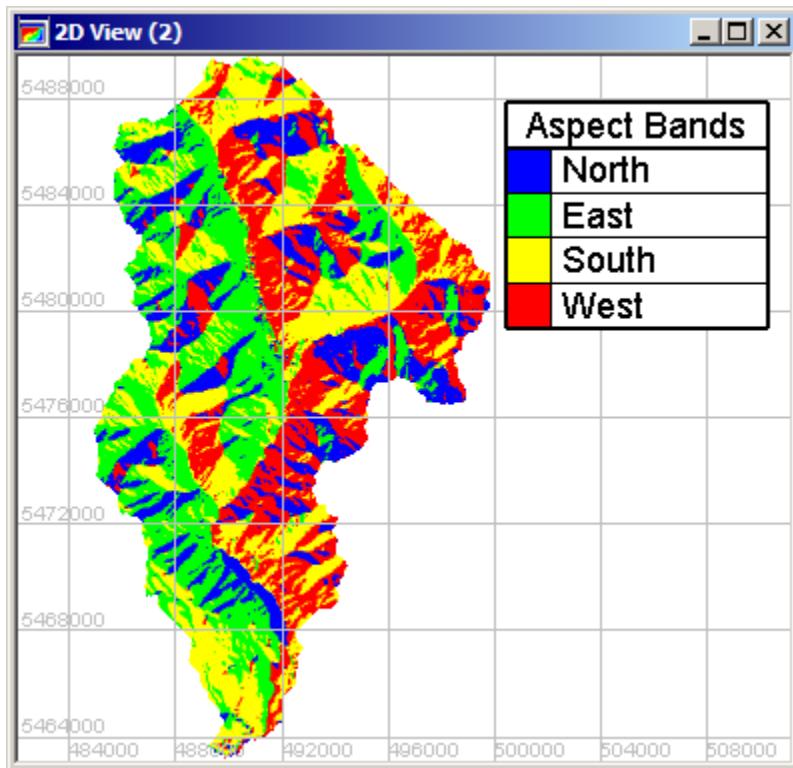


Figure 5.13: This watershed has been divided into north-, east-, south-, and west-facing aspects.

5.2.2.6 Identifying Zones Within HBV-EC

Two of the tabs within the HBV-EC panel obtain their data from user-supplied information, instead of deriving it from the watershed object. The Climate and Land Use tabs require you to provide information on their respective subjects. This information must be mapped onto the triangular mesh corresponding to each data tab. This can be done using polygons that outline each area, by importing a GeoTIFF that describes the zones, or, for Climate Zones, by using a point set to produce Thiessen polygons.

To identify a zone using polygons:

Note: These instructions assume that you do not already have a data item describing the area or perimeter of the zone to be identified. If you have such an object, open it and start with step 5.

1. Drag the map (either **Climate Zones** or **Land Use Regions**) that corresponds to the type of zone you're identifying to a 2D view window.
2. With the window open and the map visible, click on the  button on the tool bar.

3. Using the Polygon tool, trace the outline of the zone. See "Drawing Lines and Closed Polylines", on p. 69 for more information on using this tool.
4. When you've completely outlined the area, press <Esc> or click  to stop drawing. Give the line a name and click .
5. Within the WorkSpace, select the map (again, either **Climate Zones** or **Land Use Regions**) and select **Tools→Map Object...** from the menu bar.
6. From the dialog box, select the object that identifies the zone. If you've just created the object, it will have the name that you selected in Step 4. Click .
7. If the zone you're identifying is a climate zone, you're finished. The area you've defined will be added to the list of climate zones. You'll identify the properties of each climate zone on the **Simulation** panel.

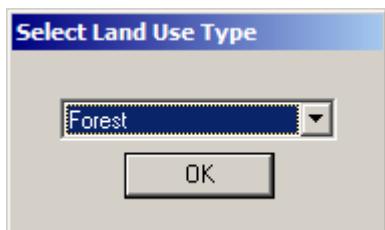


Figure 5.14: This dialog lets you choose a Land Use Type

If you're identifying a land use region, a dialog box will appear, letting you select which type of region you're describing. Select the land use category and click . You'll see the area on the map change colour to match the new land use type.

In either case, you can check the **Area** column on the appropriate tab on the **Basin** panel to confirm that the changes have been applied.

To identify a zone using a GeoTIFF:

If you have a GeoTIFF that describes the climate zones or land use regions, the process is similar to using polygons.

Note: If you're describing land use regions, the GeoTIFF **must** have no more than four classes numbered from 0 to 3, corresponding to the land use classes used by HBV-EC. Undescribed areas will retain the default value of 0, for Open land. See "The Land Use Tab", on p. 245 for more information on these classes, and see "Classification of a GeoTIFF Image", on p. 130 for more information on reclassifying a GeoTIFF..

1. From the menu bar, select **File→Import→GeoTIFF** and select the GeoTIFF that you're using.
2. Within the WorkSpace, select the mesh corresponding to the map you're identifying, either **Climate Zones** or **Land Use Regions**, and select **Tools→Map Object...** from the menu bar.
3. From the dialog box, select the GeoTIFF that identifies the zone. Click . The areas of the map will automatically be assigned to categories based on the value of the corresponding area on the GeoTIFF.

To identify a climate zone using points:

1. Create or load a Point Set object with a point at the centre of each of the climate zones to be identified. See "Drawing Points", on p. 68 for more information on creating a point set.
2. In the WorkSpace, right-click on the Climate Zones object and select **Map Climate Zones from points** from the shortcut menu.

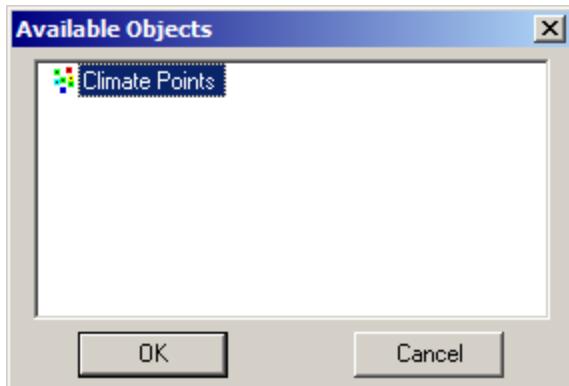


Figure 5.15: The Climate Points point set can be selected from this dialog

3. From the **Available Objects** dialog, select the point set that corresponds to the climate zones. Click **OK**.

Each point on the Climate Zones map will be assigned a value equal to that of the point to which it is closest. The number of values assigned to the members of the point set determine the number of climate zones on the map. You can identify the properties of each climate zone on the **Simulation** panel.

5.2.3 The Simulation Panel

The purpose of the **Simulation** panel is to provide information to detail the physical characteristics of each of the areas that have been identified on the **Basin** panel.

Because the HBV-EC model is one-dimensional, the relative locations of the different types of terrain are not relevant. Only the characteristics that have been described on the Basin and Simulation panels are considered. For example, one "area" might consist of all forested terrain located in Climate Zone 1, within Elevation Band 3, with a slope between 2 and 8%, facing north. A change in any of the five defining characteristics would result in a particular portion of terrain falling into a different area.

As a result of this approach, the total number of areas is equal to the number of climate zones multiplied by the number of elevation bands, multiplied by the number of land use types appearing, multiplied by the number of slope bands, multiplied by the number of aspect bands. Note, though, that **Lake** terrain is always considered to have a **Slope** and **Aspect** of 0.

For example, if we have one climate zone, four elevation bands, **Open**, **Forest**, and **Lake** terrain, three slope bands, and two aspect bands (**North** and **South**), the HBV-EC model will have to be executed for each of 52 areas, each of which must also be described.

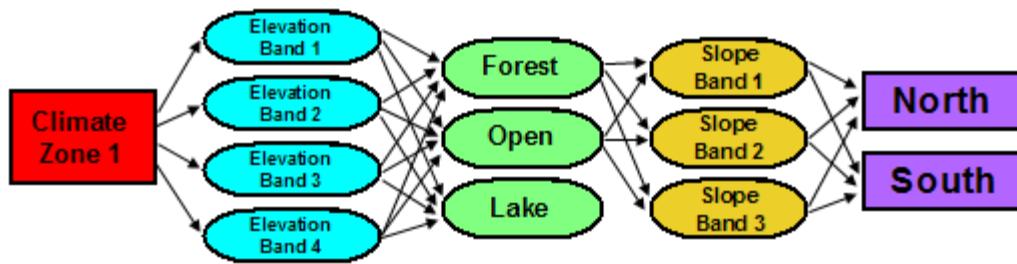


Figure 5.16: Even a small number of groups in each category can result in a large number of areas to be modelled

To activate this panel, click the **Generate Model from Spatial Basin Data** button. This will create a **Climate Zone** panel for each climate zone defined on the **Basin** panel.

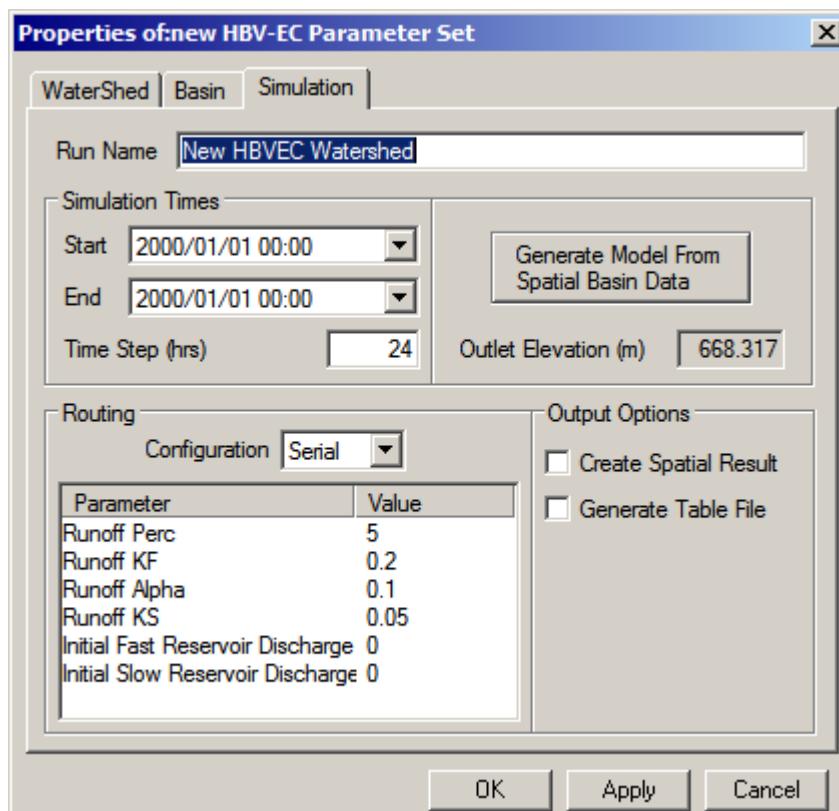


Figure 5.17: This Simulation panel has not been activated and contains no information

Note: If you use the **Basin** panel to add or remove any areas later on, you will need to regenerate the model. This will overwrite any existing information on the **Simulation** panel.

Any of the variables on this panel can be edited by clicking on the value and entering a new value, with the exception of the Outlet Elevation, which is obtained from the Watershed.

- **Run Name:** This contains the name of the watershed and of the model. The default value is **New HBVEC Watershed**.
- **Simulation Times:**

- **Start** - This is the start date and time of the simulation. In most cases, the start and end dates will be limited by the dates for which you have weather data available. Enter the date and time of the start of the simulation in the format **YYYY/MM/DD HH:MM**. The default is **2000/01/01 00:00** (midnight, January 1st, 2000). If your weather data doesn't contain data for intervals smaller than a day, use midnight (00:00) as the start time.
- **End** - This is the end date and time of the simulation. Like the Start Date, this value may be limited by available data. Enter the end date and time of the simulation as **YYYY/MM/DD HH:MM**. The default value is **2000/01/01 00:00**. Notice that this is the same as the default Start Date.
- **Time Step (hrs)** - This contains the number of hours in each step of the model simulation. The default value is **24** hours, or 1 day per step.
- **Outlet Elevation**: This is the elevation of the outlet from the watershed, in metres. This value is determined from the basin of the watershed, and cannot be edited.
- **Routing**: These variables apply to the entire watershed, regardless of the number of land classes or climate zones.
 - **Configuration** - This parameter selects the model to be used to calculate The value for this variable can be either **Parallel** or **Serial**. If the value is set to **Parallel**, the **Runoff FRAC** variable will be available. If it is set to **Serial**, the **Runoff Perc** variable will be available. The default value is **Serial**.
 - **Runoff Perc** - This is the rate of percolation from the fast reservoir to the slow reservoir, per day. This simulates the effects of groundwater recharge on the slow reservoir. This variable is only available if the **Routing Model** is set to **Serial**. The default value is **5**.
 - **Runoff FRAC** - This is the fraction of runoff directed to the fast reservoir. Watersheds that respond quickly to precipitation will tend to have higher values, while watersheds that show a delayed response will have lower values. This variable is only available if the **Routing Model** is set to **Parallel**. The default value is **0.7**.
 - **Runoff KF** - This is the fast reservoir coefficient, which determines what proportion of the fast reservoir is released per day. At a value of **0**, the fast reservoir won't release any water, while at a value of **1**, it will empty itself each day. The default value is **0.2**.
 - **Runoff Alpha** - This is the fast reservoir exponent. In conjunction with **Runoff KF**, it determines the release rate of the fast reservoir. The default value is **0.1**.
 - **Runoff KS** - This is the slow reservoir coefficient. Like **Runoff KF**, it determines the amount of the slow reservoir released each day, but for the slow reservoir. The default value is **0.05**.
 - **Initial Fast Reservoir Discharge** - This is the rate of discharge from the fast reservoir at the beginning of the simulation, in thousands of litres per second. The default value is **0**.
 - **Initial Slow Reservoir Discharge** - This is the rate of discharge from the slow reservoir at the beginning of the simulation, in thousands of litres per second. The default value is **0**.
- **Output Options**: These options determine what output will be generated by the Simulation when the model is run.

- **Create Spatial Result:** If this option is checked, the model will generate the various 2D output data as described under "The Results of the HBV-EC Model", on p. 260.
- **Generate Table File:** If this option is checked, the simulation will generate an **.hbt** (HBV-EC Output Tableset) file when run. This file is created with the same name and in the same location as the **.hbv** file.

5.2.4 The Climate Zone Panel

The variables shown on this panel are specific to a particular climate zone. In most cases, an HBV-EC simulation will contain only a single climate zone, and so there will only be a single panel. If the simulation contains multiple climate zones, the variables for each will be shown on a separate tab, named **Climate Zone 1**, **Climate Zone 2**, and so on..

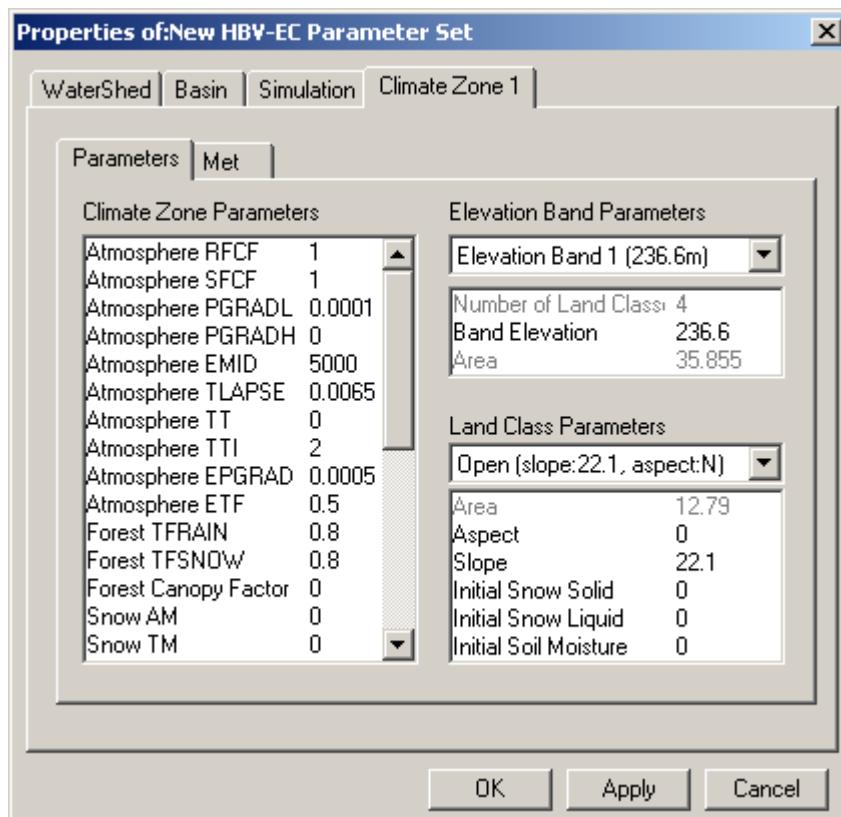


Figure 5.18: These variables are specific to a particular Climate Zone

5.2.4.1 The Parameters Tab

The variables listed on this panel are specific to a single climate zone, as defined on the **Climate** tab of the **Basin** panel, and identified on the **Climate Zones** mesh.

- **Climate Zone Parameters**

These variables apply to all land classes within a particular climate zone.

- **Atmosphere** variables

- **RFCF** - This is the rainfall correction factor. This parameter allows you to account for potential error in the recorded rainfall values, due to gauge undercatch or known differences between the location of the measuring station and the location of the simulated area. This must be a positive number, and the default value is **1**.
- **SFCF** - This is the snowfall correction factor. It's essentially the same as **RFCF**, above, but applies to snowfall measurements, instead of rainfall. The default value is **1**.
- **PGRADL** - This is the fractional increase in precipitation with elevation, for elevations below **EMID**, per metre. This parameter allows for the orographic effects of altitude on precipitation levels. The default value is **0.0001**, which indicates a 0.01% increase in precipitation per metre of elevation above the measuring station. This value, and **PGRADH**, below, must fall between **0** and **1**.
- **PGRADH** - This is the fractional increase in precipitation with elevation, for elevations above **EMID**, per metre. This value can be used when there's a difference in orographic effects above and below a certain level. The default value is **0**, which indicates that elevations above **EMID** no longer experience an increase in precipitation.
- **EMID** - This is the mid-point elevation separating precipitation gradients, in metres. The default value is **5000**. Below this elevation, **PGRADL** is used to determine the precipitation increase from elevation; above it, **PGRADH** applies.
- **TLAPSE** - This is the temperature lapse rate, in degrees Celsius per metre. The default value is **0.0065**. This parameter takes into account the elevation of the MET file data when calculating the effective temperature at a particular elevation. Typically, this value should fall between the saturated and adiabatic temperature lapse rates, so values from **0.006** to **0.010** can be expected.
- **TT** - This is the threshold air temperature for distinguishing rain from snow, in degrees Celsius. The default value is **0**. This value represents the midpoint of the range determined by **TTI**, below.
- **TTI** - This is the temperature interval for mixed rain and snow, in degrees Celsius. The default value is **2**. Temperatures less than **TTI** degrees on either side of **TT** will experience mixed precipitation, while temperatures below **TT-TTI** will be entirely snow, and above **TT+TTI** will be entirely rain.
- **EPGRAD** - This is the fractional rate of decrease of potential evaporation with elevation, per metre. The default value is **0.0005**.
- **ETF** - This is the temperature anomaly correction of potential evapotranspiration. The default value is **0.5**. The model determines daily evaporation rates by looking at the monthly rate, and taking into account the difference between the recorded daily temperature and the normal monthly temperature. An increase in this value will increase the effect of a temperature variance on the evaporation rate. The value must fall between **0** and **1**.
- **Forest** variables
 - **TFRAIN** - This is the fraction of rainfall reaching ground surface below the forest canopy. This value must be between **0** and **1**, and the default value is **0.8**.

- **TFSNOW** - This is the fraction of snowfall reaching ground surface below the forest canopy. This value must be between **0** and **1**, and the default value is **0.8**.
- **Canopy Factor** - This is the proportion of sunlight blocked by the forest canopy. This value must be between **0** and **1**. The default value is **1**, indicating that sunlight is completely blocked by the canopy.
- **Snow** variables
 - **AM** - This is the factor controlling the influence of the aspect on the melt factor. The default value is **0**.
 - **TM** - This is the threshold temperature for snowmelt, in degrees Celcius. The default value is **0**. Lower temperatures will encourage snowmelt, while higher temperatures will discourage it.
 - **CMIN** - This is the value of the melt factor on the winter solstice for open areas, in millimetres per degree Celsius, per day. This represents the minimum value for the snow melt factor over the course of the year. The default value is **2**.
 - **DC** - This is the increase in melt factor between winter and summer solstices, in millimetres per degree Celcius, per day. The default value is **2**. The sum of **CMIN** + **DC** gives the snowmelt factor on the summer solstice, the high point for the year.
 - **MRF** - This is the ratio between the melt factor in forest to the melt factor in open areas. The default value is **0.7**. At a value of 1.0, snow will melt as easily in a forest as on open ground. Typically, this value will range between **0.6** and **1.0**, depending on the forest type, coverage, and the age of the forest.
 - **CRFR** - This controls the rate at which liquid water refreezes in snowpack, in millimetres per degree Celsius, per day. The default value is **2**. This represents the opposite of the process controlled by **CMIN** and **DC**.
 - **WHC** - This is the liquid water holding capacity of snowpack, expressed as a fraction of snowpack water equivalent. The default value is **0.05**, which indicates that snowpack can consist of 5% liquid water before it begins to flow.
 - **LWR** - This is the maximum amount of liquid water that can be retained by a snowpack, in millimetres. The default value is **2500**. This value comes into play when the snowpack is extremely deep.
- **Soil** variables
 - **FC** - This is the field capacity of the soil, in millimetres. The default value is **200**. This indicates the maximum amount of water that the soil can retain.
 - **BETA** - This controls the relationship between soil infiltration and soil water release. The default value is **1**. Values less than this indicate a delayed response, while higher values indicate that runoff will exceed infiltration.
 - **LP** - This is the soil moisture content (as a proportion of **FC**) below which evaporation becomes supply-limited, meaning that actual evaporation will be less than potential evaporation. This value must be between **0** and **1**, and the default value is **0.7**.
- **Glacier** variables

- **MRG** - This is the ratio of melt of glacier ice to seasonal snow at the same air temperature. The default value is **2** and the minimum is **1**. There is no maximum value. This account for the difference in albedo between ice and snow, resulting in a greater amount of energy being needed to melt the same amount of ice as snow.
- **AG** - This is the factor controlling the relation between glacial snowpack water equivalent and runoff coefficient, per millimetre. This simulates the effect of sub-glacial drainage systems. The default value is **0.05**. The value is typically between **0** and **0.2**.
- **DKG** - This is the difference between the minimum and maximum outflow coefficients for glacier water storage, per day. The default value is **0.05**. The maximum value would be typical of a mature glacier in late summer.
- **KGMin** - This is the minimum outflow coefficient for glacier water, per model time step. The default value is **0.05**. This value would indicate an early spring outflow coefficient.
- **KGRC** - This is a recession coefficient that is applied to the computation of the glacier outflow coefficient. This value must be between **0** and **1**, and the default value is **0.7**.
- **Area** - This is the total area contained within the climate zone, in square kilometres. This value is obtained from the **Basin** panel, and cannot be edited.
- **Elevation Band Parameters**

These variables deal with a specific **Elevation Band**. Since most watersheds will contain at least a few such bands, you may need to provide some information for each of them.

To edit the variables for an **Elevation Band** other than the first, select the band you'd like to edit from the list box shown at the top of Figure 5.19.

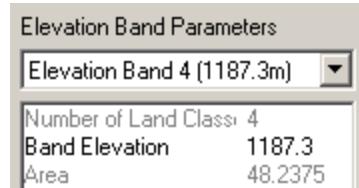


Figure 5.19: The Elevation Band information contains only one editable variable

- **Number of Land Classes** - This variable shows the number of Land Classes contained within the Elevation Band. A **Land Class** consists of a particular combination of **Land Use Region**, **Slope Band**, and **Aspect Band**. Remember, Lakes are always considered to have a **Slope** and **Aspect** of **0**, so all lakes within a particular **Elevation Band** are considered to comprise a single area. This value is obtained from the **Basin** panel, and cannot be edited.
- **Band Elevation** - By default, this variable gives the median elevation for the Elevation Band. All areas within the band are considered to have this elevation. If this value isn't appropriate, for example, if the majority of terrain is near the top of the band, and hence, above the median, enter the corrected value here.

- **Area** - This variable shows the total area of the terrain contained within the **Elevation Band**, in square kilometres. This value is obtained from the **Basin** panel, and cannot be edited.
- **Land Class Parameters**

A **Land Class** consists of a specific combination of **Land Use Region**, **Slope Band**, and **Aspect Band**. All of the Land Classes within the particular **Elevation Band** chosen in the previous area will be shown in the list displayed at the top of Figure 5.20.

Note: Because all **Lake** terrain is considered to have a **Slope** and **Aspect** of **0**, with no snow, no soil and no canopy, the only variable listed for **Lake** terrain is **Area**.

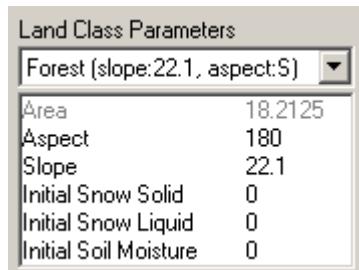


Figure 5.20: These variables apply to a specific Land Class

- **Area** - This variable displays the total area of the selected **Land Class**. This value is obtained from the **Basin** panel, and cannot be edited.
- **Aspect** - This variable shows the bearing of the **Land Class**, in degrees, with **0°** indicating North, **90°** indicating East, and so on. All terrain within the class will be considered to have this bearing. If the value is inappropriate, you can change this variable.
- **Slope** - This is the **Slope** of the **Land Class**, in degrees. All terrain within the class will be considered to have this slope. If the value is not appropriate, you can change this variable.
- **Initial Snow Solid** - This is the initial snow solid, in millimetres. This value represents the total liquid water content of the snow pack, or the amount of water that would be obtained if the solid portion of the snow were completely melted. The default value is **0**.
- **Initial Snow Liquid** - This is the initial snow liquid water content, which represents the amount of liquid water found within a sample of snow. The default value is **0**.
- **Initial Soil Moisture** - This is the initial moisture content of the soil, as a proportion. This variable is only listed for **Open** or **Forest** terrain. The default value is **0**.

5.2.4.2 The Met Tab

This tab describes the HBM file associated with a particular climate zone. The HBM file contains meteorological information for a period of time, including monthly average temperature and evaporation rate, and daily rainfall, snowfall, and temperature measurements. For more information on HBM files, see "The HBV-EC HBM File", on p. 327. The Met tab is broken down into five sections.

- **Display:** The HBM file data may be displayed as a 1D time series. This panel controls the display settings of the data. See "Display Properties" under Properties of Data Items, on p. 17 for more details.

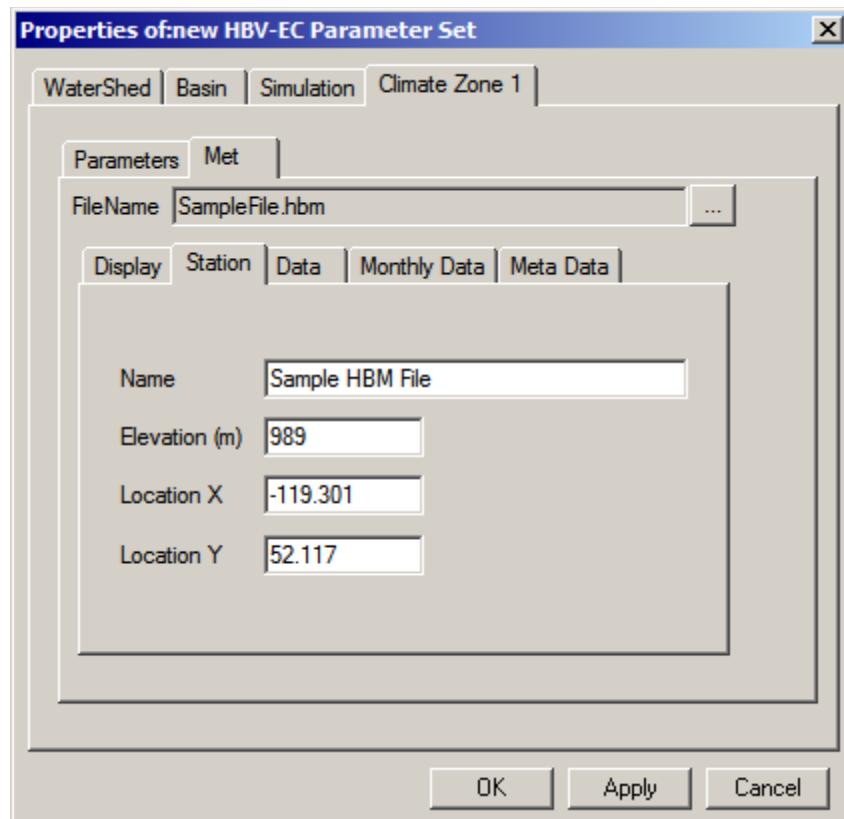


Figure 5.21: The Station data can be edited

- **Station:** This tab shows data from the header of the HBM file. This data may all be edited on this panel.
 - **Name:** This is the name of the climate station.
 - **Elevation (m):** This is the elevation of the climate station, in metres. This value is taken into account when calculating variations in rainfall, snowfall, or temperature as a result of altitude changes, as determined by the **Atmosphere** parameters on the **Parameters** panel.
 - **Location X:** This is the horizontal location of the climate station. This value is not used by the model, since HBM files are assigned to climate zones, and need not be located within the zone.
 - **Location Y:** This is the horizontal location of the climate station. This value is also not used by the model.
- **Data:** This tab shows the Start and End dates of the meteorological data, as well as the total number of records and the time step between records. It also shows the variables that are contained within the file, their maximum and minimum values, their units of measure, and which variable is currently active. None of these values can be edited directly. See "Data Attributes" under Properties of Data Items, on p. 21 for more details on this tab.

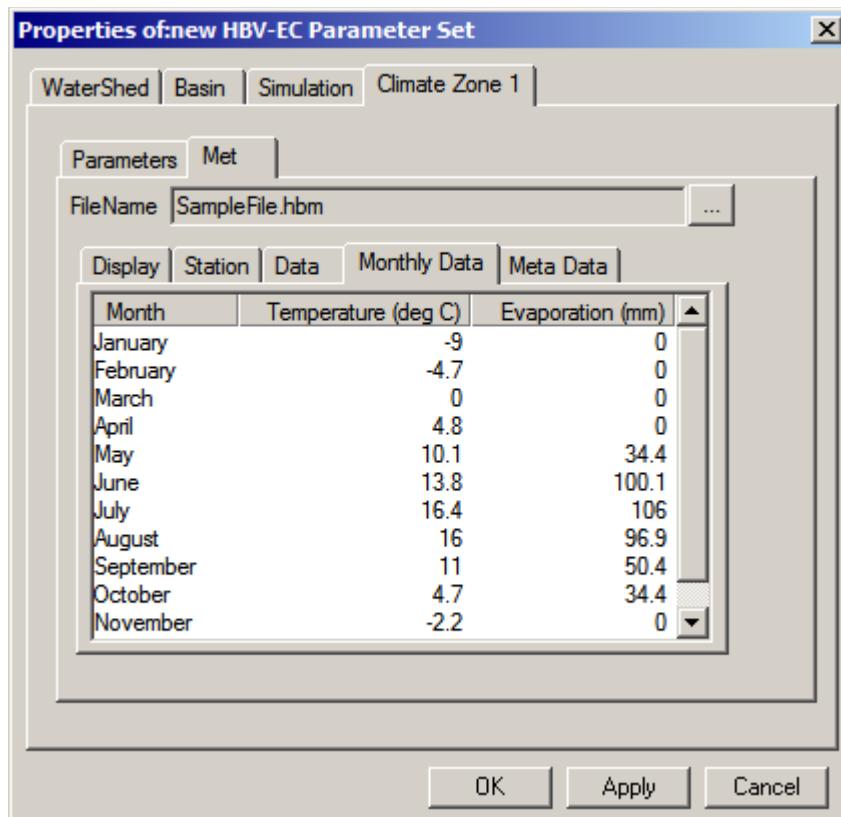


Figure 5.22: These values show the monthly averages over several years

- **Monthly Data:** The average monthly values for temperature and evaporation are displayed on this tab, and can be edited.
- **Meta Data:** This tab shows any data located in the header of the HBM file that isn't displayed on one of the other tabs. This includes the path and name of the file, the version number of the software used to create the file, and so on. See "Meta Data" under Properties of Data Items, on p. 31 for more information on meta data.

5.3 THE HBV-EC MODEL

When the HBV-EC model is run, the information supplied in the HBV-EC dialog is used to calculate several results, including both one- and two-dimensional data objects.

To run the HBV-EC model:

1. After you've entered all of the required information in all three panels of the HBV-EC dialog, click **Apply** to confirm your entries.
2. Within the WorkSpace, select the name of your **HBV-EC Parameter Set** object. From the menu bar, select **File→Save**.

3. Select **Run→Check Parameters** from the menu bar. A dialog box will appear, listing any errors remaining in the parameter set. If there are no errors found, the message box shown in Figure 5.23 will appear.

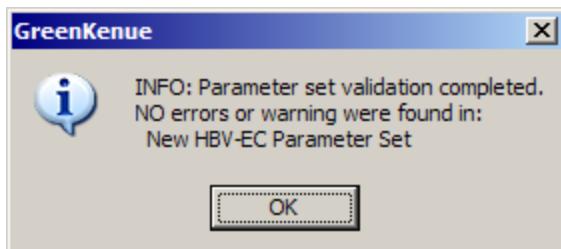


Figure 5.23: This message indicates that no errors were found

4. Once you have fixed any errors, select **Run→Launch Simulation** from the menu bar. A number of data objects will be created as children of the parameter set.

5.3.1 The Results of the HBV-EC Model

The results of the HBV-EC model are shown as 1-dimensional time series and attributes of a single 2-dimensional time-varying triangular mesh. All of these result data objects are shown in the WorkSpace as children of the HBV-EC Parameter Set.

The following results are shown as 1-dimensional time series. They are best examined in a 1-D view window.

- **Total Discharge** - This object shows the rate of total discharge, in thousands of litres per second, from the watershed over the simulated period.
- **Fast Reservoir Storage** - This object shows the level of storage in the fast reservoir of the watershed, in millimetres, over the simulated period.
- **Slow Reservoir Storage** - This object shows the level of storage in the slow reservoir of the watershed, in millimetres, over the simulated period.
- **Fast Reservoir Discharge** - This object shows the rate of discharge from the fast reservoir of the watershed, in thousands of litres per second, over the simulated period.
- **Slow Reservoir Discharge** - This object shows the rate of discharge from the slow reservoir of the watershed, in thousands of litres per second, over the simulated period.
- **Glacier Discharge** - This object shows the rate of discharge from glaciers within the watershed, in thousands of litres per second, over the simulated period.
- **Glacier Ice Melt** - This object shows the ice melt from glaciers within the watershed, in mm, over the simulated period.
- **Glacier Water Storage** - This object shows the water storage from glaciers within the watershed, in mm, over the simulated period.

The following results are shown as attributes of a time-varying 2-dimensional triangular mesh. These results are generated only when the **Create Spatial Result** checkbox on the **Simulation** panel of the HBV-EC Model parameters dialog is active. They are best examined in a 2D view

window, and can be animated to see the changes of the data over time. For more information on examining time-varying data, see "Animation", on p. 61.

- **Temperature** - This attribute shows the temperature of the air for each altitude band, in degrees Celsius, over the simulated period.
- **Rainfall** - This attribute shows the rainfall, in millimetres per simulation time step, for each area over the simulated period.
- **Snowfall** - This attribute shows the snowfall, in millimetres per simulation time step, for each area over the simulated period.
- **Soil Moisture** - This attribute shows the soil moisture, in millimetres, for each area over the simulated period.
- **Soil Infiltration** - This attribute shows the soil infiltration, in millimetres, for each area over the simulated period.
- **Water Release** - This attribute shows the water release, in millimetres, for each area over the simulated period.
- **Evaporation** - This attribute shows the evaporation rate, in millimetres per simulation time step, for each area over the simulated period.
- **Snow Water Equivalent** - This attribute shows the snow water equivalent, in millimetres, for each area over the simulated period.
- **Glacier Ice Melt** - This attribute shows the ice melt from glaciers, in millimetres, for each area over the simulated period.
- **Glacier Water Storage** - This attribute shows water storage from glaciers, in millimetres, for each area over the simulated period.

APPENDIX A: FILE TYPES OF ENSIM CORE

General Information

This appendix, about file types supported by EnSim, is included for two reasons:

- To allow you to create or edit files with external applications so that they may be used within EnSim.
- To allow you to become familiar with the file types used by EnSim.

There are a few basic object types native to EnSim. Each is represented in the WorkSpace by a particular icon.

- **2D Rectangular Grid:** scalar  and vector 
- **2D Triangular Mesh:** scalar  and vector 
- **Line Set:** 2D  and 3D 
- **XYZ Point Set, Point Set, or Parcel Set:** 
- **XY Data Item:** 
- **Time Series:** scalar  and vector 
- **Table:** 
- **Velocity Roses:** 
- **Network:** 
- **2D Rectangular Cell Grid:** 

These icons help to identify an object's type. Many objects can retrieve and save information from and to several file formats. For example, the source data of a 2D line set object might be an i2s, a shp, or an mif file. Even though these files have different formats, the underlying data, once imported into EnSim, is handled in the same way. See "Supported Foreign File Formats [EnSim Core]", on p. 304, for more information about file types from other software packages.

File Headers

All EnSim native file formats have similar headers. The first portion of the header is identical for all of these files. An example (from an xyz file) is shown below.

```
#####
:FileType xyz ASCII EnSim 1.0
# Copyright (c) Canadian Hydraulics Centre/National Research Council 1998-2005
# DataType XYZ Point Set
#
:Application GreenKenu
:Version 3.1.33
:WrittenBy Username
:CreationDate Fri, Apr 15, 2005 11:20 AM
#
```

- *Keywords* begin with the colon character (:). Keywords may have specific meanings within the context of the application file type, such as the origin, count, and delta of a rectangular grid. When EnSim recognizes one of the keywords, it will look for the proper information immediately following. Keywords that have no direct meaning within the context of a particular file type are treated as Meta Data. Keywords can be added to the file in the **MetaData** tab of the **Properties** dialog by using the **+** button, or can be added directly into the file by using a text editor.

Note: The **:EndHeader** keyword must be the last line of the header. Other keywords can appear in any order, but if a given keyword appears more than once in a header, the last listed appearance will replace any earlier occurrences.

Standard keywords appearing in the top section of the header (like those shown above) are as follows:

- **:FileType** - Required - Shows the file type in the form of the file extension, such as r2s, i3s, t3s, and so on; whether the file is ASCII or binary; and the version number of the EnSim file type. This tells EnSim the format of the data in the file, based on its type.

Note: A binary file still has an ASCII header so that it can be read in a text editor.

- **:Application** - Optional - States the EnSim Application, such as Green Kenu, Blue Kenu, or WaveSim, with which the file was created.
- **:Version** - Optional - This gives the version of the EnSim Application with which the file was created.
- **:WrittenBy** - Optional - This is the username assigned to the computer workstation or account with which the file was created.
- **:CreationDate** - Optional - This is the date and time at which the file was last altered.

Other keywords that are not specific to a particular file type and are included below the dashed line of the header include:

- **:Name** - Optional - The default is the root of the filename. It is the name of the object that is displayed in the WorkSpace. The entry may be changed directly in the file header, or by editing the **Name** field of the **MetaData** tab of the object's **Properties** dialog.

Note: If you save an object using the **Save As...** command, the **:Name** keyword will be stripped from the file. When the file is loaded, the **Name** field on the **MetaData** tab will be set to the file name of the object.

- **:Title** - Optional - The default for this keyword is the full file path of the file. The entry for this keyword is displayed in the title bar of the object's **Properties** dialog.
- **:Projection** - Optional - The coordinate system of the data. Valid values are:
 - **Cartesian** - Cartesian coordinates.
 - **LatLong** - Latitude/Longitude coordinates. Must appear with the **:Ellipsoid** keyword.
 - **UTM** - UTM coordinates. Must appear with the **:Zone** and **:Ellipsoid** keywords.
 - **MTM** - MTM coordinates. Must appear with the **:Zone** and **:Ellipsoid** keywords
 - **PolarStereographic** - Polar Stereographic coordinates. Followed by **:CentreLongitude**, **:CentreLatitude** and **:Ellipsoid** keywords.
 - **LambertConformal** - Lambert Conformal coordinates. Followed by **:CentralMeridian**, **:FirstStandardParallel**, **:SecondStandardParallel**, **:LatitudeOfOrigin**, **:FalseEasting** and **:FalseNorthing** keywords.
 - **Albers** - Albers coordinates. Followed by **:CentralMeridian**, **:FirstStandardParallel**, **:SecondStandardParallel**, **:LatitudeOfOrigin**, **:FalseEasting** and **:FalseNorthing** keywords.

The default value is **Cartesian**. A data item with no **:Projection** keyword may be opened, but it will be assigned the **Cartesian** coordinate system when it is saved.

- **:Zone** - Required if the **:Projection** is **UTM** or **MTM**. This keyword contains an integer from 1 to 60 for **UTM** coordinates, or from 1 to 32 for **MTM** coordinates.
- **:Ellipsoid** - Required if **:Projection** is present and is not **Cartesian** - The ellipsoid used by the coordinate system. Default is **Unknown**. Possible values are **Clark66**, **GRS80**, **WGS72**, **WGS74**, or **Sphere**.
- **:CentreLatitude** - Required if the **:Projection** is **PolarStereographic** - The centre latitude of the projection. A negative centre latitude indicates a Southern Polar Stereographic projection.
- **:CentreLongitude** - Required if the **:Projection** is **PolarStereographic** - The centre longitude of the projection.
- **:CentralMeridian** - Required if the **:Projection** is **LambertConformal** or **Albers** - The central meridian is the longitude of the centre of the projection. It is also referred to as the **Longitude of Origin**.

- **:LatitudeOfOrigin** - Required if the **:Projection** is LambertConformal or Albers - The latitude of origin is the latitude where the central meridian crosses the projection origin or base line.
- **:FirstStandardParallel** - Required if the **:Projection** is LambertConformal or Albers - The first standard parallel is the latitude nearest the equator where the cone of the conic projection intersects the globe.
- **:SecondStandardParallel** - Required if the **:Projection** is LambertConformal or Albers - The second standard parallel is the latitude nearest the pole where the cone of the conic projection intersects the globe.
- **:FalseEasting** - Required if the **:Projection** is LambertConformal or Albers - The false easting is the value added to the x coordinate. It is usually used to remove negative coordinate values.
- **:FalseNorthing** - Required if the **:Projection** is LambertConformal or Albers - The false northing is the value added to the y coordinate. It is usually used to remove negative coordinate values.
- **:AttributeName** - Required or Optional, depending on file type - Together with **:AttributeUnits** and **:AttributeType**, this keyword identifies data attributes associated with the data in the file.

The data following this keyword is in two parts. The first part is an integer number. It refers to the order of the attribute with respect to the other attributes in the file. If there is one attribute, the number is 1. If there are three attributes, the first to be read is 1, the second is 2, and so on. The second part of the keyword is a name used to identify the data attribute. For example, **:AttributeName** 1 Elevation.

- **:AttributeUnits** - Optional - Together with **:AttributeName** and **:AttributeType**, this keyword identifies data pertaining to data attributes associated with data points.

The data following this keyword is in two parts. The first part is an integer number. If there is a single attribute, this number is 1. If there are three attributes, the first to be read is numbered 1, the second 2, and so on. The second part of this keyword is a text entry to identify the units of the data attribute, such as m, g/L, km**2, and so on. For example, **:AttributeUnits** 1 m.

- **:AttributeType** - Required or Optional, depending on file type - Together with **:AttributeName** and **:AttributeUnits**, this keyword identifies data pertaining to data attributes associated with data points.

This keyword is followed by an integer and text. The integer identifies the placement of the attribute with respect to the order of the other attributes. The number 3 would mean that this attribute is the third in the list of attributes to be read from the data in the file. The text following the integer identifies the type of attribute data. Attribute types are:

- float
- integer
- boolean

- text
- date
- oneof

In the file data, the text or date must be enclosed by double quotation marks ("). If this keyword is not used, the attribute type is not specified and the attribute is given the default type of floating point (float).

If the attribute type `oneof` is used, a list of character strings (words) enclosed in double quotes must follow on the same line as the type declaration. In the body of the file, integers are used to index these character strings. For example, a 3 in the file body that corresponds to an attribute with the type `oneof` would be assigned the third character string on the list on the `:AttributeType` keyword line.

- `:SourceFile` - Optional; only used with data extraction files - States the name of the file from which the data was extracted.
- `:FrameTime` - Optional; only used with data extraction files - States the simulation time at which the data was extracted. Applicable to data extraction files carrying data from one frame of a parent file containing time-varying data.
- `:EndHeader` - Required - Always the last keyword in a header. Anything appearing after this keyword is considered part of the body of the file.

Comments begin with the pound character (#). All characters following # on a line are ignored. The `Copyright` comment is the same for each file. The `DataType` comment depends on the type of file. It corresponds to the type of file specified by the keyword `:FileType`, but written out in full for informational purposes. The example below shows a file type keyword line and a data type comment line from the same file:

```
:FileType i3s ASCII EnSim 1.0  
# DataType 3D Line Set
```

Other header information varies depending on the type of file. However, all headers **must** end with the keyword `:EndHeader`.

ASCII and Binary Files

Files containing data that does not vary with time may be in either ASCII or binary format. Files having data that does vary over time must be in binary format. Both ASCII and binary files have ASCII headers. This file attribute is specified in the file header with the keyword `:FileType`. After the three-letter file extension, the identifier `ASCII` or `BINARY` is written.

ASCII Files

The format of ASCII files is usually fairly simple, and they can be edited and saved with any text editor. The data delimiter in ASCII files is white space; in most cases, any number of spaces, tabs, or line returns may separate the data. The formats of ASCII files vary, and depend on the type of data.

The time-related content of the file header may be in the form `yyyy/mm/dd hh:mm:ss` or the ISO standard format of `yyyy-mm-dd hh:mm:ss`. A relative time stamp in an ASCII file, must be in the format `hhhhhh:mm:ss.mm`, and may have a value of up to 2,147,483,647 hours.

Binary Files

In binary files, the values, or attributes, associated with the object geometry may change at each time step. Each step therefore follows a similar format that is repeated for the total number of steps. The data at each time step is referred to as a *record*.

Each record has a header that specifies the time in the simulation at which the data was recorded. All *record headers* have the same format, regardless of the file type.

The record header consists of nine 4-byte integer variables defined as follows:

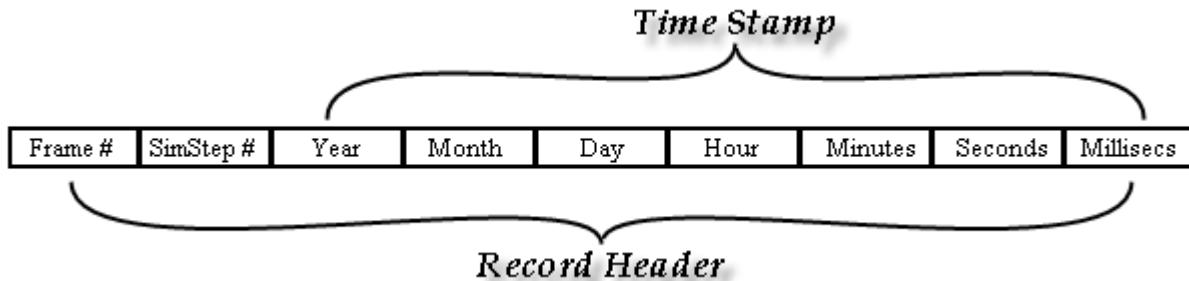


Figure A.1: The record header can record both placement within an animation and absolute time of a record

1. **Frame number:** Increases sequentially with each record. The first record must have a value of 1, the second a value of 2, and so on.
2. **Simulation time step number:** Increases sequentially with each record. The values may increase at any increment, and may vary by more than one increment through the file.
3. **Year**

The remaining integer values contain the time stamp, which records the simulation date and time.

4. **Month**
5. **Day**
6. **Hour**
7. **Minute**
8. **Second**
9. **Millisecond**

If the year and month are zero, the time stamp is considered to be an arbitrary time. If the year and month are non-zero, the time stamp is a date.

The remaining content of a binary file depends on the data type.

NATIVE FILE FORMATS

2D Rectangular Grids [r2s / r2v]

Two-dimensional rectangular, or regular, grids have orthogonal evenly-spaced data points connected by straight lines. The grid may be georeferenced. The surface created by the connected points lies in the xy plane of a Cartesian coordinate system.

The file extensions for 2D rectangular grids are ***.r2s** and ***.r2v**. Their icons are  and  respectively. The **s** in r2s indicates that the file contains scalar data and similarly, the **v** in r2v indicates vector data. Both r2s and r2v files may contain time-varying data, which is always stored in a binary format. Non-time-varying files may be saved as ASCII or binary data. The scalar rectangular grid (r2s) can have one or more data attributes.

R2s files may be saved as any of the following formats:

- ArcINFO ASCII Grid - ***.asc**
- Surfer Grid - ***.grd**
- 2D Triangular Mesh - ***.t3s** or ***.t3v** (see p. 274)
- XYZ Point Sets - ***.xyz** (see p. 281)
- 2D Rectangular Cell - ***.r2c**
- GeoTIFF Image - ***.tif**, if all values within the file are between 0 and 255.

File Headers [r2s / r2v]

The contents of the header is similar for each type of rectangular grid. A sample header from a non-time-varying grid is shown below.

```
#####
:FileType r2s  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType          2D Rect Scalar
#
:Application        GreenKenue
:Version            3.1.33
:WrittenBy          Username
:CreationDate       Fri, Apr 15, 2005 11:20 AM
#
#-----
#
:Name   Jock River Grid
:Title  Jock River Grid
#
:Projection  Cartesian
:Ellipsoid Unknown
#
:xOrigin 407778.859400
:yOrigin 4975883.000000
#
:AttributeName 1 Elevation
```

```

:AttributeUnits 1 m
:xCount 100
:yCount 100
:xDelta 400.000000
:yDelta 400.000000
:Angle 0.000000
#
:EndHeader

```

Note: The header for a binary file, such as one containing data that varies over time, is similar, except that the third field in the `:FileType` record is `BINARY`, instead of `ASCII`. If the file contains vector data, the `:FileType` will be listed as `r2v` and the `:DataType` as `2D Rect Vector`.

For more information on header keywords, see "File Headers", on p. 264.

- `:xorigin` - Optional - This is the x-coordinate of the point in the bottom left corner of the grid. The default value is 0.
- `:yorigin` - Optional - This is the y-coordinate of the point in the bottom left corner of the grid. The default value is 0.
- `:xCount` - Required - The number of points, or vertices, in each row of the grid, along the x-direction.
- `:yCount` - Required - The number of points, or vertices, in each column of the grid, along the y-direction.
- `:xDelta` - Required - The distance between two adjacent points in a row.
- `:yDelta` - Required - The distance between two adjacent points in a column.
- `:Angle` - Optional - The clockwise angle of rotation, in degrees, of the grid about the origin, or bottom-left corner. The default is 0.

Data Organization [r2s / r2v]

The ordering of points on a rectangular grid begins at the bottom-left (southwest) corner of the grid and proceeds to the right along the bottom row. When the end of the row is reached, the numbering resumes at the left end of the next row up. The diagram below shows a simple r2s grid.



Figure A.2: The points of this grid are numbered sequentially left to right and bottom to top

Keywords in the header are used to indicate the coordinates of the origin of the grid, the spacing between the vertices, the number of points in the x- and y-directions of the grid, and the angle of the grid. From this information, the coordinates of each point on the grid are determined.

File Formats [r2s / r2v]

The file format refers to the organization of data that is contained in the file after the `:EndHeader` keyword; that is, after the entire file header.

ASCII

Only non-time-varying data can be stored in ASCII format.

Data values are recorded in *free format*. This means that any number of spaces, tabs, or line returns may separate individual values. The first value in the data set corresponds to the first point of the grid, the second value to the second point, and so on. The data is read until a value is obtained for each grid point, so the number of values in the file must correspond to the total number of vertices in the grid: `xCount * yCount`. If there are more values than grid points, only the number of values equal to the number of grid points will be used, and excess values will be ignored. In the case of more than one attribute, once the data for each node is read for the first attribute, then the data for the next attribute is expected starting again at the first node 1. Each attribute will be read sequentially.

If the file has been saved by an EnSim application, the values at each grid point will be written in a matrix format. The number of columns in the file corresponds to the number of vertices in the x-direction of the grid, and the number of rows corresponds to the number of vertices in the y-direction.

For scalar data, *.r2s, the first value listed in the matrix is the value at the bottom-left corner of the rectangular grid. The matrix of values is repeated for each successive attribute

For vector data, *.r2v files, the matrix is the same except that all of the x-components of each vector for a particular data attribute are listed first. After the x-components for each point are listed, the y-components of the vectors are listed. A vector data set contains twice as many values as a single attribute scalar data set with the same number of grid points.

Binary

Both time-varying and non-time-varying data may be stored in binary format. However, the header of the binary file is stored as ASCII characters so that the file can be examined with a text editor.

For each time step, new values must be specified for each point on the grid. There will be a new data record for each time step. The format given below is repeated for each time step.

The first numbers in the data record comprise a record header that specifies the time step, data and time of the record. See Figure A.1 on p. 268 for more information on the record header.

For **Scalar** data, each record header is followed by a sequential collection of sub-records representing the values for each node of the grid for each data attribute. Each data attribute sub-record stores n values, where n is the total number of nodes in the grid. Each value is a 4-byte floating point number. The values for each node are listed in order, beginning at zero index.

RH1	V1 of A1	V2 of A1	...	Vn of A1	V1 of A2	V2 of A2	...	Vn of A2	V1 of An	V2 of An	...	Vn of Ap
...												
RH2	V1 of A1	V2 of A1	...	Vn of A1	V1 of A2	V2 of A2	...	Vn of A2	V1 of An	V2 of An	...	Vn of Ap
...												
RHm	V1 of A1	V2 of A1	...	Vn of A1	V1 of A2	V2 of A2	...	Vn of A2	V1 of An	V2 of An	...	Vn of Ap

- **RH:** Record Header, numbered from 1 to m . Each frame has it's own record header.
- **V:** Node value. $V1$ is the value of the first node, and Vn is the value of the last node. The order of these values corresponds to the order of the points of the grid. See "Data Organization [r2s / r2v]", on p. 271 for more information on the ordering of points within a grid..
- **A:** Attribute. $A1$ is the first attribute, Ap is the last. Note: The number of attributes is determined from the :AttributeName keywords in the file header.

The above grid illustrates the layout of a binary r2s file for a grid containing n nodes, p attributes, and data varying over m frames. Each scalar value is a 4-byte floating point number.

For **Vector** data, there are two data sets contained within the file. The first gives the x-component of the vector at each point of the grid, and the second gives the y-component of each point. The order of the values in each block of data corresponds to the order of points in the grid, as given in "Data Organization [r2s / r2v]", on p. 271.

Note: Vector files are single attribute.

RH1	X at N1	X at N2	...	X at Nn	Y at N1	Y at N2	...	Y at Nn
RH 2	X at N1	X at N2	...	X at Nn	Y at N1	Y at N2	...	Y at Nn
...								
RHm	X at N1	X at N2	...	X at Nn	Y at N1	Y at N2	...	Y at Nn

Similar to the previous grid, this grid illustrates the layout for a binary r2v file containing n nodes varying over m frames. Each x- and y-component is a 4-byte floating point value.

2D Triangular Meshes [t3s / t3v]

The three (3) in t3s stands for 3-node triangles, the simplest possible finite element grid. The data contained in a triangular mesh file is in two sections: the first lists the coordinates and the data attributes of the nodes, and the second lists the connectivity of the nodes making up the elements. The file extensions for a 2D triangular mesh are ***.t3s** and ***.t3v**. Their icons are  and  respectively. The 's' in t3s indicates that the data in the file are scalar and the 'v' in t3v indicates that the data in the file are vector.

Both t3s and t3v files may contain time-varying data, which is always stored in binary format. Non-time-varying data may be stored in ASCII or binary format. The scalar triangular mesh (t3s) can have one or more data attributes.

T3s files can be exported into the following formats:

- Trigrid Neigh - *.ngh
- Trigrid Node - *.nod
- GoogleEarth Keyhole Markup Language - *.kml

File Headers [t3s / t3v]

An example of an ASCII file for a simple mesh containing scalar data that does not vary over time is given below:

```
#####
:FileType t3s  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2004
# DataType          2D T3 Scalar Mesh
#
:Application      BlueKenue
:Version          3.0.44
:WrittenBy        UserName
:CreationDate    Fri, April 15, 2005 11:20 AM
#
#-----
#
:Projection  Cartesian
:Ellipsoid Unknown
#
:NodeCount 17
:ElementCount 24
:ElementType  T3
#
:EndHeader
7.855000 57.107000 14
23.788315 39.900479 0
25.561000 22.693000 1
31.297000 65.835000 0
33.791000 35.411000 4
38.571468 29.716530 0
39.526000 84.539000 22
48.753000 56.110000 9
52.993000 27.681000 7
```

```
55.654222 79.650735 0
62.718000 71.322000 8
65.461000 52.369000 3
66.659458 70.452329 0
68.794803 27.252671 0
73.394006 72.916187 0
76.933000 32.419000 6
96.883000 47.631000 5
1 2 4
1 4 7
1 3 2
2 3 5
2 5 8
2 8 4
3 14 9
3 9 6
3 6 5
4 8 10
4 10 7
5 6 8
6 9 8
8 9 12
8 12 11
8 11 10
9 14 12
10 11 15
11 12 13
11 13 15
12 14 16
12 16 17
12 17 15
12 15 13
```

For more information on header keywords, see "File Headers", on p. 264.

- **:NodeCount** - Required - The total number of nodes in the mesh.
- **:ElementCount** - Required - The total number of triangular elements in the mesh.
- **:ElementType** - Optional (Currently, the default, T3, is the only supported value) - Shows the type of finite element used in the mesh. EnSim only creates triangular finite element meshes with three nodes per element, called T3 meshes.

File Formats [t3s / t3v]

The data contained in a triangular mesh file is divided into two parts: the first lists the coordinates and the values of the data attributes of each mesh point, and the second part lists the node indices of the elements, or the connectivity of the mesh.

ASCII

Only files that contain non-time-varying data can be stored in ASCII format.

The coordinates and values of the mesh points are listed in order by node number. The first set of coordinates, on the first line of data, applies to node 1. The second set, on line 2, applies to node 2, and so on. There is a coordinate and value set for each node of the mesh. In the above

example, there are 17 entries, since the `:NodeCount` keyword has a value of 17. The first number on a line is the x-coordinate, the second number is the y-coordinate, and the third and following numbers are the values of the data attributes. In the example, node 1 has coordinates of (7.855000, 57.107000) and has a single value of 14 (this is a single attribute file).

Scalar data has one or more values following the coordinates. Each value represents an attribute.

Vector data has two data values following the coordinates. The first is the x-component of the vector and the second is the y-component.

The connectivity of the mesh is listed below the coordinate and value data. Each line in the connectivity section lists the node numbers of the nodes that comprise the vertices of a single triangular element. Each element is listed on a separate line. In the example, there are 24 elements, and hence 24 lines of data.

The format of a triangular mesh file is slightly stricter than that of a rectangular grid. The information for each node or element must begin on a new line, and there cannot be any blank lines between nodes or elements. Optionally, there may be a blank line between the list of coordinates and the list of element nodes. All of the information for a particular line or element must appear on the same line, but there may be any number of spaces between values on a particular line.

Binary

Both time-varying and non-time-varying meshes may be recorded in binary format. However, the header of the binary file is left in ASCII characters so that it can be examined in a text editor.

The binary file begins with a section that describes the mesh, including the coordinates of the nodes and the connectivity of the nodes forming the elements. There are five records which contain this data.

The first two records contain the x- and y-coordinates of the nodes. Each of these contains n floating point values, where n is the total number of nodes in the mesh, as indicated by the `:NodeCount` keyword. The first position in the record refers to node 1, the second to node 2, and so on.

The next three records specify the node connectivity. There are m integer values in each of these records, where m is the total number of elements, or triangles, in the mesh, as indicated by the `:ElementCount` keyword. The first record contains the values for the first node in each element. The next two hold the values for the second and third nodes in each element, respectively.

X at N1	...	X at Nn	Y at N1	...	Y at Nn	N1 of E1	...	N1 of Em	N2 of E1	...	N2 of Em	N3 of E1	...	N3 of Em
---------	-----	---------	---------	-----	---------	----------	-----	----------	----------	-----	----------	----------	-----	----------

- **X:** X-coordinate (4-byte floating point)
- **Y:** Y-coordinate (4-byte floating point)
- **N1:** Node 1 of the mesh (4-byte integer)

- **Nn:** Node n of the mesh (the last node) (4-byte integer)
- **E1:** Element 1 of the mesh (4-byte integer)
- **Em:** Element m of the mesh (the last element) (4-byte integer)

Following the description of the mesh, which does not vary with time, are the values of the data attributes, which may vary with time. Since the values at each mesh point may change with time, each time step must list a new set of mesh values.

Each data record begins with a record header. See "Binary Files" under ASCII and Binary Files, on p. 268 for more information on record headers. After the header, each record contains the values of the data attribute at that particular time step.

For **Scalar** data, each record header is followed by a sequential collection of sub-records representing the values for each node of the mesh for each data attribute. Each data attribute sub-record stores n values, where n is the total number of nodes in the mesh. Each value is a 4-byte floating point number.

RH1	V1 of A1	V2 of A1	...	Vn of A1	V1 of A2	V2 of A2	...	Vn of A2	V1 of An	V2 of An	...	Vn of Ap
...												
RH2	V1 of A1	V2 of A1	...	Vn of A1	V1 of A2	V2 of A2	...	Vn of A2	V1 of An	V2 of An	...	Vn of Ap
...												
RHm	V1 of A1	V2 of A1	...	Vn of A1	V1 of A2	V2 of A2	...	Vn of A2	V1 of An	V2 of An	...	Vn of Ap

- **RH:** Record Header, numbered from 1 to m. Each frame has it's own record header.
- **V:** Node value. V1 is the value of the first node, and Vn is the value of the last node.
- **A:** Attribute. A1 is the first attribute, Ap is the last. Note: The number of attributes is determined from the :AttributeName keywords in the file header.

The above table illustrates the layout of a binary triangular mesh containing n nodes, p attributes, and data varying over m frames..

For **vector** data, there are two records in the file, each sufficient to hold n 4-byte floating-point values, where n is the total number of nodes in the mesh. The first record holds all of the x-components of the vector. The second holds all of the y-components. The values in the records are stored in the sequence of the nodes. The first value refers to the first node, the second value to the second node, and so on.

RH1	X at N1	X at N2	...	X at Nn	Y at N1	Y at N2	...	Y at Nn
RH2	X at N1	X at N2	...	X at Nn	Y at N1	Y at N2	...	Y at Nn
...								
RHm	X at N1	X at N2	...	X at Nn	Y at N1	Y at N2	...	Y at Nn

The above table illustrates the node values for a mesh containing n vector nodes, varying over m time frames. Each value is a 4-byte floating point number.

Line Sets [i2s / i3s]

Line set files consist of one or more 2D or 3D lines and may have additional attributes associated with each line. Line sets include lines, opened polylines, and closed polylines. File extensions for line sets are ***.i2s** and ***.i3s**, for 2-dimensional and 3-dimensional line sets, respectively.

- A 2-dimensional line set has vertex geometry that is defined by x- and y-coordinates. Its icon in the WorkSpace is .
- A 3-dimensional line set has vertex geometry defined by x-, y-, and z-coordinates. Its icon in the WorkSpace is . In many cases, the z-coordinate is elevation.

Both i2s and i3s files may also include data attributes, but each attribute must be equal at all vertices within a single line set.

Both 2D and 3D line sets may be saved in the following formats:

- 2D lines - ***.i2s**
- 3D lines - ***.i3s**
- Point data - ***.xyz**
- ArcView Shape - ***.shp**
- MapInfo Interchange - ***.mif**
- GoogleEarth Keyhole Markup Language - ***.kml** (LatLong only)

File Headers [i2s / i3s]

An example of a 2D line set file is shown below.

```
#####
:FileType i2s ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2004
# DataType 2D Line Set
#
:Application BlueKenue
:Version 3.0.44
:WrittenBy Username
:CreationDate Fri, Apr 15, 2005 11:20 PM
#
#-----
#
:Projection Cartesian
:Ellipsoid Unknown
#
:AttributeName 1 fract1
:AttributeType 1 float
:AttributeName 2 name
:AttributeType 2 text
:AttributeName 3 subname
:AttributeType 3 text
:AttributeName 4 fract2
```

```
:AttributeType 4 float
:AttributeName 5 whole
:AttributeType 5 integer
:EndHeader
4 0.3 "Southern Ontario Lakes" "Ontario" 0.5 3
21.563981 84.360190
2.606635 49.289100
44.075829 24.170616
90.047393 45.023697

6 0.4 "Southern Ontario Lakes" "Erie" 0.75 2
-8.473834 94.596351
-18.269154 19.145908
11.911023 14.115879
-0.531682 33.177043
-2.384851 57.532975
6.086778 79.506262
```

Line set files have no file-specific keywords. See "File Headers", on p. 264 for information on general keywords.

File Formats [i2s / i3s]

ASCII

All line set files are in ASCII, as currently only non-time-varying line sets have been implemented. The following format is repeated for each line in the set.

The first line of data contains at least one value. The first of these is an integer that represents the total number of vertices in the line. The other values are the attributes associated with the line. The number, name, and type of attributes are specified by the `:AttributeName` and `:AttributeType` keywords in the file header.

The following lines of data contain the x- and y-coordinates (and the z-coordinate, in the case of a 3D line set) of each vertex in the order in which they are connected. If the data represents a closed polyline, or polygon, the coordinates on the first and last lines must be identical. The first number in each line is the x-coordinate, the second number is the y-coordinate, and the third number, if present, is the z-coordinate.

Binary

Because EnSim does not support time-varying line sets, there is no binary format for i2s or i3s files.

XYZ Point Sets [xyz]

XYZ Point sets are stored in a relatively simple format. They contain data describing points that are not connected, and may not follow any particular pattern. Their data does not vary over time, and contains only a single attribute. The icon representing point sets in the WorkSpace is . The file extension of a point set file is ***.xyz**.

File Headers [xyz]

An example of a point set file is shown below.

```
#####
:FileType xyz ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2004
# DataType XYZ Point Set
#
:Application BlueKenu
:Version 3.0.44
:WrittenBy Username
:CreationDate Fri, Apr 15, 2005 11:20 AM
#
#-----
#
:EndHeader
39.526000 84.539000      22
27.855000 57.107000      14
25.561000 22.693000      1
96.883000 47.631000      5
48.753000 56.110000      9
62.718000 71.322000      8
76.933000 32.419000      6
33.791000 35.411000      4
31.297000 65.835000      0
65.461000 52.369000      3
52.993000 27.681000      7
```

Point sets have no unique keywords. Only general keywords are used. See "File Headers", on p. 264 for more information on general keywords.

File Format [xyz]

Point sets have one of the simplest file formats of any EnSim file type. Data is always stored in ASCII format. The data in a point set file is organized into rows of coordinates that position individual points in space. Each line holds the coordinates for a different point. The first number on a particular line is the x-coordinate of the point, the second number is the y-coordinate, and the third is the z-coordinate. If no z-coordinates are given in the file, all z-coordinates are considered to have a default value of 0. EnSim can also read files that do not contain a header, but do contain properly formatted data; that is, files with two or three values per line. If the file is then saved within EnSim, a header will be added.

Note: XYZ ASCII files may also be comma delimited.

XY Data Objects [xy] [dat]

XY data objects are stored in a relatively simple format. Their data does not vary over time. XY data objects are, however, very useful for comparing two data attributes. The icon used to represent an XY data item in the WorkSpace is . The file extension of an XY data item may be ***.xy** or ***.dat**.

File Headers [xy] [dat]

An example XY data file is shown below.

```
#####
:FileType xy  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType          XY DataObject
#
:Application        GreenKenuue
:Version            3.1.33
:WrittenBy          Username
:CreationDate       Fri, Apr 15, 2005 11:20 AM
#
#-----
#
:AttributeName 1
:AttributeUnits 1
:AttributeName 2
:AttributeUnits 2
:EndHeader
2853.12 2099.55
2506.79 1861.46
2507.27 1861.46
2172.7 1629.13
2922 2007.38
2512.55 1851.86
2513.03 1851.86
2513.03 1852.34
2136.7 1712.17
2137.18 1712.65
```

Like point sets, XY data objects have no unique keywords. See "File Headers", on p. 264 for more information on general keywords.

File Format [xy] [dat]

XY data objects have a very simple layout. Data is always stored in ASCII format, organized into two columns. Each column represents a single attribute, such as discharge, level, width, and so on. EnSim is capable of reading files without a header, as long as the data portion of the file is properly formatted; that is, with two values per line. If the file is then saved from within EnSim, a header will be added. This format is particularly useful for illustrating the relationship between two attributes, such as Discharge vs. Level, as found in a rating curve analysis.

Parcel Sets [pcl]

Parcel sets contain point data possessing multiple attributes. Since their contents are displayed as points, parcel sets are represented by the point set icon in the WorkSpace: . They use the file extension ***.pcl**. The data of a parcel set may vary over time. If the data does not vary over time, the file is stored in ASCII. If it does vary over time, the file is in binary format. Parcels are only referenced in two dimensions (x and y). However, elevation may be added as one of the attributes.

Parcel sets may be saved in ASCII in the following formats:

- Multi-frame MapInfo Interchange - *.mif
- Single-frame MapInfo Interchange - *.mif
- Single-frame Parcel - *.pcl

With the multi-frame *.mif option, all frames are saved in a single time step. When the data is displayed, all frames appear simultaneously. With single-frame *.mif and parcel files, only the frame that is being displayed when the file is created is saved.

File Headers [pcl]

An example parcel set file is shown below

```
#####
:FileType pcl  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType          Lagrangian Parcel Set
#
:Application      GreenKenuue
:Version          3.1.33
:WrittenBy        Username
:CreationDate    Fri, Apr 15, 2005 11:20 AM
#
#-----
#
:AttributeName 1 State
:AttributeUnits 1 NONE
:AttributeName 2 Thickness
:AttributeUnits 2 metres
:AttributeName 3 Volume
:AttributeUnits 3 litres
:AttributeName 4 Radius
:AttributeUnits 4 metres
:AttributeName 5 Volume Fraction
:AttributeUnits 5 Percent
:AttributeName 6 Exposure Time
:AttributeUnits 6 sec.
#
:EndHeader
18871.355469 22274.814453 1 9.43762e-006 28.0892 30.7797 0.280892 61321
19500.916016 23334.943359 1 4.05161e-006 27.1164 46.1559 0.271164 61321
19521.322266 23338.529297 1 4.57495e-006 25.1464 41.8283 0.251464 61321
18809.814453 22344.748047 1 1.19806e-005 29.8996 28.1851 0.298996 61321
```

```

18685.777344 21950.718750 1 5.67973e-006 24.3049 36.9069 0.243049 61321
18251.394531 21331.871094 1 2.72589e-006 23.6394 52.5398 0.236394 61321
18806.742188 22367.779297 1 1.48637e-005 32.0508 26.1988 0.320508 61321
18636.828125 22065.134766 1 2.51945e-006 24.6362 55.7903 0.246362 61321
18491.363281 22131.228516 1 2.70968e-006 25.6871 54.9318 0.256871 61321

```

File Formats [pcl]

ASCII

Parcel files that contain data that does not vary over time are stored in ASCII format. The data is organized into $n+2$ columns, where n is the total number of data attributes possessed by each point. The two extra columns are used to store the x- and y-coordinates of the points. The information for each point begins on a new line. The first value on a line is the x-coordinate, and the second is the y-coordinate. Data attributes follow on the same line, in the same order as they appear in the header.

Binary

Parcel files containing data that varies over time are stored in binary format. Each time step is contained in a separate record that describes the location and attributes of each point at that particular time step.

The first item in each record is the record header. See "Binary Files", on p. 268 for more information on record headers.

After the record header is an integer indicating the number of points, or parcels, present at the particular time step. That number is followed by the x-coordinates of all of the points, which is followed by the y-coordinates for all of the points.

After the coordinates, the data attributes are listed. Each attribute is contained within a subrecord, which contains all of the values for that attribute at that time step. The listing of attribute values is followed by a repetition of the number of points in the time step, which ends the record.

Points, or parcels, can appear in any order, as long as that order is maintained throughout the file.

RH1	#P	X of P1	...	X of Pn	Y of P1	...	Y of Pn	V1 of A1	...	Vn of A1	V1 of An	...	Vn of An	#P
...														
RHn	#P	X of P1	...	X of Pn	Y of P1	...	Y of Pn	V1 of A1	...	Vn of A1	V1 of An	...	Vn of An	#P

- **RH:** Record Header - RH1 indicates the header for time step 1; RHn indicates the header for the last time step recorded in the file
- **X:** X-coordinate of a point
- **Y:** Y-coordinate of a point

- **P:** Point - P1 indicates the first point in a file; Pn indicates the last point. #P indicates the total number of points recorded
- **A:** Attribute - A1 indicates the first attribute in a file; An indicates the last attribute

Point Sets [pt2]

Point sets, like parcel sets, contain point data possessing multiple attributes. Their contents are displayed as a series of multi-attribute points, represented by the point set icon in the WorkSpace: . They use the file extension ***.pt2**. The data of a point set does not vary over time, and is stored in ASCII. Points are only referenced in two dimensions (x and y). However, elevation may be added as an attribute.

Point sets may be saved as the following:

- MapInfo Interchange - ***.mif**
- ArcView Shape - ***.shp**
- XYZ point set - ***.xyz**

When the file is saved as a MapInfo Interchange or ArcView Shape file, all attributes are retained. When it is saved as an XYZ point set file, only the attribute that is currently selected when the file is saved will be recorded.

Multi-attribute MapInfo Interchange and ArcView Shape files are treated as point set files within EnSim.

File Headers [pt2]

An example point set file is shown below

```
#####
:FileType pt2 ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType Point Set
#
:Application GreenKenu
:Version 3.1.33
:WrittenBy Username
:CreationDate Fri, Oct 15, 2005 11:20 AM
#
#-----
#
:AttributeName 1 State
:AttributeUnits 1 NONE
:AttributeName 2 Thickness
:AttributeUnits 2 metres
:AttributeName 3 Volume
:AttributeUnits 3 litres
:AttributeName 4 Radius
:AttributeUnits 4 metres
:AttributeName 5 Volume Fraction
:AttributeUnits 5 Percent
#
:EndHeader
18871.355469 22274.814453 1 9.43762e-006 28.0892 30.7797 0.280892
19500.916016 23334.943359 1 4.05161e-006 27.1164 46.1559 0.271164
19521.322266 23338.529297 1 4.57495e-006 25.1464 41.8283 0.251464
18809.814453 22344.748047 1 1.19806e-005 29.8996 28.1851 0.298996
```

```
18685.777344 21950.718750 1 5.67973e-006 24.3049 36.9069 0.243049
18251.394531 21331.871094 1 2.72589e-006 23.6394 52.5398 0.236394
18806.742188 22367.779297 1 1.48637e-005 32.0508 26.1988 0.320508
18636.828125 22065.134766 1 2.51945e-006 24.6362 55.7903 0.246362
18491.363281 22131.228516 1 2.70968e-006 25.6871 54.9318 0.256871
```

File Formats [pt2]

ASCII

Point set files never contain data that varies over time. Consequently, the data are always stored in ASCII format. The data is organized into $n+2$ columns, where n is the total number of data attributes possessed by each point. The two extra columns are used to store the x- and y-coordinates of the points. The information for each point begins on a new line. The first value on a line is the x-coordinate, and the second is the y-coordinate. Data attributes follow on the same line, in the same order as they appear in the header.

Time Series [ts1 / ts2 / ts3 / ts4 / ts5]

There are five types of time series supported by EnSim: ***.ts1**, ***.ts2**, ***.ts3**, ***.ts4**, and ***.ts5**. The first four types cover the range of combinations of scalar and vector data with simple time steps or explicit date and time. They have the icons  or , for time series containing scalar or vector data, respectively.

- ts1: scalar data with simple (implicit) time steps
- ts2: vector data with simple time steps
- ts3: scalar data with explicit time (date, hours, minutes, seconds)
- ts4: vector data with explicit time

The fifth time series is a special type of time series called a timegrid. It is generated using the Extract Time Series tool - **Along a Line...** and is essentially a collection of attribute values extracted along a line from a time-varying grid or mesh. See "Extracting Time Series", on p. 96 for more information.

File Headers [ts1 / ts2 / ts3 / ts4]

An example header from a time series file is shown below.

```
#####
:FileType ts1  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2004
# DataType          Type 1 Time Series
#
:Application        BlueKenu
:Version            3.0.44
:WrittenBy          Username
:CreationDate       Fri, Apr 15, 2005 11:20 PM
#
#-----
#
:AttributeUnits 1 M
:AttributeName 1 FREE SURFACE
#
:LocationX        444997.593750
:LocationY        5022443.000000
#
:StartTime         0:00:00.000
:DeltaT            0:15:00.000
#
:EndHeader
```

- **:LocationX** - This keyword is used in time series files that were created by EnSim as a result of a time series extraction from another data tile. This value represents the x-coordinate of the location of the data probe from which the series was extracted.
- **:LocationY** - This keyword is also used in time series files that were created by EnSim as a result of a time series extraction from another data tile. This value represents the y-coordinate of the location of the data probe from which the series was extracted.

- **:StartTime** - This is the explicit time in hours:minutes:seconds.decimal seconds when the time series begins. If the type of time series uses explicit time, as is the case for ts3 and ts4 files, this is the time of the first data point. If explicit time is not specified, as with ts1 and ts2 files, the time is set to 0:00:00.000.
- **:DeltaT** - This is the time step used in files that do not use explicit time, as is the case with ts1 and ts2 files.

File Headers [ts5]

An example header from a type 5 time series file is shown below.

```
#####
:FileType ts5  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2007
# DataType          Type 5 Time Series
#
:Application      Green Kenu
:Version          3.1.33
:WrittenBy        Username
:CreationDate    Mon, Jul 16, 2007 11:20 AM
#
#-----
#
:AttributeName 1
:AttributeUnits 1
:AttributeName 2
:AttributeUnits 2
#
:BeginLine 8
:Point 452534.495453 5464098.227279
:Point 452616.191608 5464098.037764
:Point 452756.638834 5464098.166487
:Point 452849.177323 5464098.955789
:Point 452849.689540 5464098.381640
:Point 452849.003346 5464098.467922
:Point 452773.573321 5464098.720075
:Point 452715.768993 5464098.248804
:EndLine
#
:EndHeader
```

- **:BeginLine** - This keyword is used in the ts5 time series file to lead off the block of points that represent the extraction line used to generate the file. The keyword value represents the number of points in the extraction line.
- **:Point** - This keyword represents a point in the extraction line. The values are the x and y coordinates of the point.
- **:EndLine** - This keyword is used in the ts5 time series file to end the block of points that represent the extraction line used to generate the file.

File Formats [ts1 / ts2 / ts3 / ts4 / ts5]

ASCII

Time series files are always recorded in ASCII format. In all four types, the data for each time step is recorded on a new line. Each file uses a slightly different format for the layout of data.

Type 1 - *.ts1

Type 1 time series have the simplest organization of the four types of time series. They contain scalar data that varies with a constant time step. There is one column of data: the values for each time step. If more than one value is present on a line, only the first will be read. The time step is constant, and is specified with the `:DeltaT` keyword in the header. If no time step is included in the file, a dialog will appear when the file is opened in EnSim that asks you to supply a time step, in seconds. Type 1 time series files without headers can be opened within EnSim. If a headerless Type 1 time series file is saved within EnSim, it will be given a header.

An excerpt from a Type 1 time series file is shown below. A portion of the header is included to show the time step, or `DeltaT` value. The time step for this data is one minute, or 60 seconds.

```
:DeltaT          0:01:00.00
#
:EndHeader
0.500000
1.000000
2.000000
4.000000
7.000000
5.000000
1.000000
```

Type 2 - *.ts2

Type 2 time series are only slightly more complex than Type 1 time series. They contain vector data that varies with a constant time step. There are two columns of data. The first contains the magnitude, or scalar value, of the data for that time step, and the second contains the direction of the vector, in compass degrees. Each row contains data about a different time step. The time step is constant, and is specified with the `:DeltaT` keyword in the header. If no time step is included in the file, a dialog will appear when the file is opened in EnSim that asks you to supply a time step, in seconds. Type 2 time series files without headers can be opened within EnSim. If a headerless Type 2 time series file is saved within EnSim, it will be given a header.

An excerpt from a Type 2 time series file is shown below. A portion of the header is included to show the `DeltaT` value. The time step for this data is 10 seconds.

```
:DeltaT          0:00:10.000
#
:EndHeader
0.000000 0.000000
51.885268 0.000000
```

```

183.427737 120.877376
238.120234 103.580318
291.955243 89.223187
344.795983 79.220052
396.791710 75.832632
448.677744 75.550902
500.610531 77.772019
553.351561 87.234322
606.818749 100.143973

```

Type 3 - *.ts3

Type 3 time series files contain scalar data that varies with an explicit time step. Each line of data has two sets of values. The first is the explicit time and the second is the corresponding scalar data value. There are four formats that may be used to specify the date and time, although only one format may be used in a particular file. Hours, minutes and seconds are always specified, regardless of the format, although the values may be zero. The date and decimal seconds are optional. If the date is omitted, hours increment beyond 24. An example of each time format is shown below:

- 2005/04/15 14:42:27:003
- 2005/04/15 14:42:27
- 0062:42:27:003
- 0062:42:27

An excerpt of data from a ts3 file is shown below. The time increments every two hours, starting at 3 AM on April 15th, 2005, as indicated by the explicit time.

```

#
:EndHeader
2005/04/15 3:00:00.000 0.003000
2005/04/15 5:00:00.000 0.001350
2005/04/15 7:00:00.000 0.001327
2005/04/15 9:00:00.000 0.001442
2005/04/15 11:00:00.000 0.001532
2005/04/15 13:00:00.000 0.001712
2005/04/15 15:00:00.000 0.001909
2005/04/15 17:00:00.000 0.002037
2005/04/15 19:00:00.000 0.002359
2005/04/15 21:00:00.000 0.004285
2005/04/15 23:00:00.000 0.006134
2005/04/16 01:00:00.000 0.004698

```

Type 4 - *.ts4

Type 4 time series files contain vector data that varies with an explicit time step. Each line of data has 3 sets of values. The first is the explicit time, the second is the corresponding magnitude, or scalar data value, and the third is the direction in compass degrees. There are four formats that may be used to specify the time, as explained in the description of Type 3 time series.

An excerpt of a Type 4 time series is given below. This time series does not follow a regular time step.

```
#  
:EndHeader  
0:00:00.000 0.000000 90  
0:00:51.885 0.000000 120  
0:03:03.427 120.877376 270  
0:03:58.120 103.580318 66  
0:04:51.955 89.223187 45  
0:05:44.795 79.220052 30  
0:06:36.791 75.832632 25  
0:07:28.677 75.550902 15  
0:08:20.610 77.772019 300  
0:09:13.351 87.234322 87  
0:10:06.818 100.143973 85  
0:11:00.241 112.868333 210  
0:11:53.614 125.383311 175
```

Type 5 - *.ts5

Type 5 time series files contain multiple scalar data that varies with an explicit time step. Each line of data has an explicit time value plus n values where n equals the point count of the extraction line. Like the ts3 time series, there are four formats that may be used to specify the date and time, although only one format may be used in a particular file. Hours, minutes and seconds are always specified, regardless of the format, although the values may be zero. The date and decimal seconds are optional. If the date is omitted, hours increment beyond 24. An example of each time format is shown below:

- 2005/04/15 14:42:27:003
- 2005/04/15 14:42:27
- 0062:42:27:003
- 0062:42:27

An excerpt of data from a ts5 file is shown below.

```
#  
:EndHeader  
1997/04/03 23:59:54.922 222.192985 222.193712 222.193705 222.193791 222.194516  
1997/04/04 00:59:54.922 222.191448 222.192171 222.192163 222.192237 222.192944  
1997/04/04 01:59:54.922 222.190000 222.190734 222.190725 222.190809 222.191537  
1997/04/04 02:59:54.922 222.189433 222.190151 222.190143 222.190219 222.190930  
1997/04/04 03:59:54.922 222.188974 222.189712 222.189707 222.189793 222.190526  
1997/04/04 04:59:54.922 222.188998 222.189740 222.189734 222.189813 222.190541
```

Binary

There are no binary time series file formats.

Tables [tb0]

Table objects contain data values organized in a tabular format. The table columns represent data attributes and the rows represent the values at each attribute index. Table objects display the table icon:  in the WorkSpace. The data of a table may or may not vary over time, and is stored in an ASCII format using the file extension ***.tb0**.

If the table data is associated with a start time and time step, then each data attribute represents a time series and the table object as a whole can be seen as a collection of time series, all with the same start time, deltaT and point count. The individual attributes can be extracted as time series objects by selecting **Extract TimeSeries...** from the shortcut menu of the table object.

Select table data attributes may be viewed in the 1D View window.

Tables may be saved as the following:

- Table - ***.tb0**
- Comma-delimited text - ***.csv** - Only column names and tabular data are saved.

File Headers [tb0]

An example header from a table file is shown below.

```
#####
:FileType ts0 ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2007
# DataType EnSim Table Data
#
:Application Green Kenu
:Version 3.1.33
:WrittenBy Username
:CreationDate Wed, Feb 28, 2007 11:20 AM
#
#-----
#
:StartTime 2003/01/01 12:00
:DeltaT 24:00:00.000
#
:ColumnMetaData
:ColumnName Temperature Rainfall Snowfall
:ColumnUnits degC mm mm
:ColumnType float float float
:EndColumnMetaData
#
:EndHeader
-2.2 0 24.4
2.6 18.3 7.5
1.8 8.6 0
2 15.9 0
2.8 0 0
0.1 0 0
-1.1 0 0
1.8 0 0
-0.4 0 0.4
```

-1.9 0 6.2

- **:StartTime** - This keyword is only present if time is associated with the table data. In this case, the first row of table data is associated with the date: 2003/01/01 and the time 12:00. The value may be written as date and time or just a time. The times below are all valid formats:
 - 2003/01/01 12:00
 - 2003/01/01 12:00:00
 - 24:00:00.000
 - 24:00:00
- **:DeltaT** - This keyword is only present if the :StartTime keyword is present. It represents the time step used and must be a time, not a date. The times below are valid formats:
 - 24:00:00.000
 - 12:00:00
 - 72:00:00
- **:ColumnMetaData** - This keyword initiates the block of keywords and values that describe the table attributes (columns). The keywords found within the block are:
 - **:ColumnName** - The column (or attribute) names. Specify the name for each column from 1 to n.
 - **:ColumnUnits** - The column (or attribute) units. Specify units as a string for each column from 1 to n. This keyword is optional.
 - **:ColumnType** - The column (or attribute) data type. Specify the data type for each column from 1 to n. Data types may be float, integer, boolean, text, or date.
- **:EndColumnMetaData** - This keyword ends the block of keywords and values that describe the table attributes (columns).

File Format [tb0]

ASCII

Table files are always stored in ASCII format. The data is organized into n columns, where n is the total number of data attributes. The attribute values for each data index begins on a new line. The first value on each line is the value for the first attribute, the second value is the value for the second attribute and so on. Each new line of data represents the next data index. If the :StartTime and :DeltaT keywords are present in the header than each data index represents a time with the first index at StartTime and the second at StartTime plus one DeltaT and so on. If StartTime and DeltaT are not present, than the data is associated with integer indexes only.

Binary

There are no binary table file formats.

Velocity Roses [vr1]

The Velocity Rose file/object contains the probabilities of occurrence of velocities binned by magnitude and direction. The global sum of the table is 1. The icon used to represent the velocity rose object in the WorkSpace is . The file extension of a velocity rose object is **.vr1**.

File Headers [vr1]

An example velocity rose file is shown below

```
#####
:FileType vr1 ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2007
# DataType Velocity Rose
#
:Application GreenKenu
:Version 3.1.33
:WrittenBy Username
:CreationDate Thu, Jul 05, 2007 03:42 PM
#
#-----
:SourceFile winddata.ts2
#
:Name Example Velocity Rose
:Title Example Velocity Rose - Velocity Rose
#
:LocationX 0.000000
:LocationY 0.000000
#
:SectorWidth 45.000000
:SpeedBinCount 8
#
# Probabilities by sector. (Table sum should be 1)
# Dir:0.0 Dir:45.0 Dir:90.0 Dir:135.0 Dir:180.0 Dir:225.0 Dir:270.0 Dir:315.0
# 0.109489 0.113607 0.114917 0.073367 0.124743 0.195209 0.138873 0.129796
# Table Sum = 1.000000
#
:EndHeader
0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000
2.500000 0.027513 0.026483 0.026577 0.026670 0.030882 0.019745 0.028355 0.022366
5.000000 0.038181 0.030788 0.040988 0.034438 0.052218 0.056710 0.048849 0.036684
7.500000 0.027793 0.029384 0.027232 0.009639 0.027606 0.068220 0.027513 0.033595
10.000000 0.012727 0.019465 0.013382 0.002620 0.010200 0.036122 0.018248 0.022272
12.500000 0.003182 0.006457 0.006270 0.000000 0.003182 0.011698 0.009826 0.011042
15.000000 0.000094 0.001029 0.000468 0.000000 0.000655 0.001778 0.003837 0.002901
17.500000 0.000000 0.000000 0.000000 0.000000 0.000000 0.000936 0.002246 0.000936
```

File Formats [vr1]

ASCII

The data is organized into n rows of data, where n is the number of speed bins; and $r+1$ columns of data where r is the number of directions. The extra column which is the lead column is used to store the velocity value for the bin.

Networks [n3s]

A network is a connected set of polylines or segments. Each segment of the network is made up of a series of 3-dimensional vertices. When displayed in a view, the points in each segment are connected to form a polyline. The interconnected polylines form the network. Each segment may have multiple attributes. Networks have the file extension ***.n3s**, and are represented in the WorkSpace with the  icon.

Networks may be saved in any of the following formats:

- EnSim Network ASCII Single Frame - ***.n3s**
- EnSim Network Binary Single Frame - ***.n3s**
- EnSim Network Binary Multiframe - ***.n3s**
- 3D Line Set - ***.i3s**
- 2D Line Set - ***.i2s**
- Point Set - ***.xyz**
- MapInfo Interchange format - ***.mif**
- ArcView Shape file - ***.shp**
- GoogleEarth Keyhole Markup Language format - ***.kml** (LatLong only)

File Headers [n3s]

An excerpt from an ASCII network file is shown below.

```
#####
:FileType n3s  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2007
# DataType          EnSim Network
#
:Application        GreenKenu
:Version            3.1.33
:WrittenBy          Username
:CreationDate       Fri, Apr 20, 2007 11:20 AM
#
#-----
#
:Name   Channels
:Title  Network
#
:Projection UTM
:Zone 17
:Ellipsoid GRS80
#
:SegmentAttributeName 1 Velocity
:SegmentAttributeUnits 1 m/s
#
:SegmentAttributeName 2 Strickler Friction
#
:NodeAttributeName 1 Surface Elevation
```

```

:NodeAttributeUnits 1 m
#
:EndHeader
:Segment 1 4 1.034 20
 544756.000000 4782662.000000 244.730000
 544767.000000 4782623.000000 244.729000
 544773.000000 4782603.000000 244.728000
 544784.000000 4782564.000000 244.728000
:EndSegment
:Segment 2 4 1.026 20
 544784.000000 4782564.000000 244.728000
 544783.000000 4782535.000000 244.525000
 544782.000000 4782507.000000 244.322000
 544781.000000 4782478.000000 244.120000
:EndSegment
...
:Segment 10 3 1.347 15
 544784.000000 4782564.000000 244.728000
 544807.000000 4782515.000000 244.630000
 544840.000000 4782503.000000 244.548000
:EndSegment
:Node 1 1 1 246.760
:Node 2 3 -1 2 10 246.762
:Node 3 2 -2 3 8 246.763
...
:Node 9 2 -8 9 244.750
:Node 10 1 -9 244.732
:Node 11 1 -10 246.678

```

- **:SegmentAttributeName** - Gives the name of an attribute possessed by a segment of the network
- **:SegmentAttributeType** - Gives the variable type, such as integer, text, and so on, for an attribute possessed by a segment of the network
- **:SegmentAttributeUnits** - Gives the units for an attribute possessed by a segment of the network
- **:NodeAttributeName** - Gives the name of an attribute possessed by a node
- **:NodeAttributeType** - Gives the variable type for an attribute possessed by a node
- **:NodeAttributeUnits** - Gives the units for an attribute possessed by a node
- **:Segment** - Identifies the beginning of data pertaining to a segment of the network
- **:EndSegment** - Identifies the end of data pertaining to a segment of the network
- **:Node** - Identifies the list of segments that connect at a particular node

File Formats [n3s]

ASCII

Data that does not vary with time is stored in ASCII format. The file excerpt above is an example of an ASCII network file. The data is divided into two sections. The first lists the values of the data attributes associated with each segment, as well as the x-, y- and

z-coordinates of each point that makes up a segment. The second part lists how many segments of the network meet at each node, which segments connect, and the order in which they are connected.

In the first section of the file, the start of information about a particular segment is marked by the keyword **:Segment**. The keyword is followed on the same line by the segment ID number, the number of points in the segment, and the values of any data attributes associated with the segment. On the lines following the **:Segment** keyword, the x-, y-, and z-coordinates of each point are listed in order from the head to the tail of the segment. The first and last points given for each segment are the coordinates of the nodes to which it is connected. The list of points for a particular segment ends with the **:EndSegment** keyword.

In the second section of the file, following the list of segments, the nodes of the network are listed, with each node marked by the keyword **:Node**. On the same line, after the keyword, the node ID number, the number of segments that meet at the node, and the ID numbers of each segment are listed. If a segment ID is preceded by a minus sign (-), the node is located at the tail of the segment. After the list of segment IDs, the values of any data attributes associated with the node are listed.

Binary

Networks containing data that varies over time is stored in binary format. The geometry of the network does not vary over time. Each segment described in the file consists of the same points, listed in the same order, although the attributes associated with the segments may change.

The header of the file and the section describing the geometry of the network are stored in ASCII format so that they can be examined with a text editor. The remainder of the file, containing the time-varying data, is in binary format. This section contains the data attributes possessed by both the segments and nodes of the network. Essentially, a binary network file is similar to an ASCII network file with time-varying binary data appended.

The binary portion of the network file contains a record of attribute values for each time step. The record header for these records follows the same format as the general record header for binary files, described in the section "Binary Files" under ASCII and Binary Files, on p. 268.

After the record header is a 4-byte integer that states the number of data attributes. Following that value is another 4-byte integer which states the number of segments in the network. These numbers are then followed by a series of records, one per data attribute. The same pattern—number of attributes, number of nodes, and values of the attributes—is used for the node attributes of the network. This record structure is repeated once for each time step of the network.

RecHeader										
#Seg Atts	#Segs	A1 of S1	...	A1 of Sn	...	An of S1	...	An of Sn	#Segs	#Seg Atts
#Node Atts	#Nodes	A1 of N1	...	A1 of Nn	...	An of N1	...	An of Nn	#Nodes	#Node Atts

- **RecHeader:** The Record Header. Each time step has a single record header.
- **#Seg Att:** The number of data attributes associated with segments of the network.
- **#Node Att:** The number of data attributes associated with nodes of the network.
- **#Seg:** The total number of segments in the network, numbered 1 to n.
- **#Node:** The total number of nodes in the network, numbered 1 to n.
- **A:** Attribute. **S:** Segment. **N:** Node.

2D Rectangular Cell Grids [r2c]

A 2-dimensional rectangular cell grid is similar to an ordinary 2D rectangular grid. The primary difference is that the cell grid contains information about the area enclosed by the lines and vertices of the grid, as opposed to information at the vertices.

The ordering of the cells in a 2D rectangular cell grid begins at the bottom left corner of the grid and proceeds to the right along the bottom row. When the end of a row is reached, the numbering resumes at the left end of the next row up.



Figure A.3: The cells in this grid are numbered from 1 to 40

Keywords in the header of the r2c file are used to indicate the coordinates of the origin of the grid, the height and width of each cell, the number of cells in the x- and y-directions of the grid, and the angle of the grid. From this information, the coordinates of each cell of the grid are determined.

File Headers [r2c]

An example of a header from an r2c grid is shown below.

```
#####
:FileType r2c ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType 2D Rect Cell
#
:Application GreenKenue
:Version 3.1.33
:WrittenBy Username
:CreationDate Fri, Apr 15, 2005 11:20 AM
#
#-----
#
:Projection Cartesian
:Ellipsoid Unknown
#
:AttributeName 1 Soil Moisture
#
:xOrigin 0.000000
:yOrigin 0.000000
#
```

```
:FrameTime 313:00:00.000
#
:xCount 27
:yCount 38
:xDelta 1.000000
:yDelta 1.000000
#
:EndHeader
```

For an explanation of the common keywords used in r2c files, see "File Headers", on p. 264. The remaining keywords used in 2D rectangular cell grid files are similar to those used in rectangular grid files.

- **:xorigin** - This is the x-coordinate of the point in the bottom left corner of the grid.
- **:yorigin** - This is the y-coordinate of the point in the bottom left corner of the grid.
- **:xCount** - The number of points, or vertices, in each row of the grid, along the x-direction.
- **:yCount** - The number of points, or vertices, in each column of the grid, along the y-direction.
- **:xDelta** - The distance between two adjacent points in a row.
- **:yDelta** - The distance between two adjacent points in a column.
- **:Angle** - The clockwise angle of rotation, in degrees, of the grid about the origin, or bottom-left corner.

An additional keyword is sometimes used in r2c files:

- **:FrameTime** - When data in an r2c grid is extracted from a data file containing time-variable information, this keyword is used to identify the time at which the data was extracted, in hours:minutes:seconds.decimal seconds.

File Formats [r2c]

ASCII

The ASCII data in an r2c file is organized into a grid. The number of columns corresponds to the number of cells in the x-direction, which is recorded as the value of the xCount keyword of the grid. The number of rows corresponds to the number of cells in the y-direction, recorded as the value of the yCount keyword. The first value of the data file corresponds to the bottom left cell of the grid. Cell numbering begins at zero. The next data value in the file, found to the right of the first value in the same row, is the value of the next cell in the r2c grid. Each value in the grid is saved as a floating point number with up to 6 decimal places.

Kenue can also read files that are not properly formatted, to some extent. The values may be in free format, with any number of spaces, tabs, or line returns between the values. The values are read from left to right, top to bottom and assigned to cells according to that order.

Multiframe ASCII

The ASCII r2c file is capable of storing time-varying data. The ASCII data for each frame is organized exactly the same as for a single frame ASCII file except the data is blocked within **:Frame** and **:EndFrame** keywords.

```
:EndHeader
:Frame 1 1 "1993/01/01 1:00:00.000"
  0.0 0.0 0.0 0.1 0.0 0.0 0.0 0.0 0.0
  0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
  0.0 0.1 0.1 0.0 0.0 0.0 0.0 0.0 0.0
  0.1 0.5 0.4 0.1 0.1 0.1 0.1 0.0 0.0
  0.1 0.3 0.5 0.4 0.4 0.1 0.1 0.0 0.0
  0.0 0.0 0.1 0.2 0.4 0.4 0.3 0.0 0.1
  0.0 0.0 0.0 0.0 0.0 0.0 0.1 0.0 0.1
  0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.1
  0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
  0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
  0.2 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
:EndFrame
:Frame 2 2 "1993/01/01 2:00:00.000"
  0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
  0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
  0.0 0.2 0.4 0.0 0.0 0.0 0.0 0.0 0.0
  0.0 0.3 0.4 0.2 0.1 0.2 0.0 0.0 0.0
  0.1 0.3 0.5 0.3 0.3 0.4 0.1 0.0 0.0
  0.1 0.3 0.5 0.5 0.4 0.5 0.3 0.1 0.2
  0.0 0.2 0.4 0.5 0.5 0.5 0.4 0.3 0.3
  0.0 0.2 0.4 0.5 0.4 0.3 0.5 0.4 0.4
  0.0 0.0 0.2 0.3 0.1 0.0 0.0 0.1 0.2
  0.0 0.0 0.0 0.0 0.2 0.0 0.0 0.0 0.0
  0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
:EndFrame
```

- **Frame**: The data for each timestep (or frame) is lead off by the Frame keyword. The values following the keyword are the integer frame number, the integer step number and the time associated by this frame. If there is a space within the time string, then the string must be enclosed by double quotation marks. The following are both valid time strings: "1993/01/01 1:00:00.000" and 12:00:00"
- **EndFrame**: The data for each timestep (or frame) ends with this keyword.

Note: On loading the ASCII multiframe r2c file, Kenue will save the ASCII file to a binary file. This is required to enable Kenue to visualize multiframe. A message will be displayed to the user.

Binary

Both time-varying and non-time-varying data can be saved in binary format. The format used is very similar to that found in time-varying rectangular grid files. The primary difference is that the values are listed for each cell, instead of each vertex. One record is saved for each time step of the data item.

The first numbers in each record comprise the record header, as described in the section "Binary Files" under ASCII and Binary Files, on p. 268.

After the record header are a sequential collection of sub-records representing the values for each cell of the grid for each data attribute. Each data attribute sub-record stores n values, where n is the total number of cells in the grid. Each value is a 4-byte floating point number. The values for each cell are listed in order, beginning at zero.

RH1	V1 of A1	V2 of A1	...	Vn of A1	V1 of A2	V2 of A2	...	Vn of A2	V1 of An	V2 of An	...	Vn of Ap
...												
RH2	V1 of A1	V2 of A1	...	Vn of A1	V1 of A2	V2 of A2	...	Vn of A2	V1 of An	V2 of An	...	Vn of Ap
...												
RHm	V1 of A1	V2 of A1	...	Vn of A1	V1 of A2	V2 of A2	...	Vn of A2	V1 of An	V2 of An	...	Vn of Ap

- **RH:** Record Header, numbered from 1 to m. Each frame has it's own record header.
- **V:** Node value. V1 is the value of the first cell, and Vn is the value of the last cell. The order of these values corresponds to the order of the cells of the grid.
- **A:** Attribute. A1 is the first attribute, Ap is the last. Note: The number of attributes is determined from the :AttributeName keywords in the file header.

SUPPORTED FOREIGN FILE FORMATS [ENSIM CORE]

EnSim supports several file types used by other applications. Most are GIS-related files used to import georeferenced data into EnSim. For information on the organization of these files, refer to the documentation from the parent application. These files can be opened by selecting **File→Import** from the menu bar in EnSim. Foreign file types that are supported by EnSim include:

- **ArcInfo ASCII grid files - *.asc, *.arc** - These files are read in as rectangular grids and can be saved as rectangular grids within EnSim. See "2D Rectangular Grids [r2s / r2v]", on p. 270 for more information.
- **ArcView Shape Files - *.shp** - Only the GIS features in point or line format, 2D or 3D, are supported by EnSim. The legends and other view decorations cannot be imported. Shape files are represented in the WorkSpace by the ArcView icon: . The data they contain, lines or points, are represented as children with an icon identifying the specific data type. Each data type may have multiple attributes. Shape files cannot possess multiple children.

When an ArcView Shape file is opened in EnSim, it may be treated as a line set, point set, XYZ point set, or parcel set, depending on the type of data it contains.

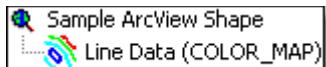


Figure A.4: An ArcView Shape file containing line data in the WorkSpace

- **NRCan and USGS Digital Elevation Maps - *.dem** - DEM files are loaded into EnSim as rectangular grids. In the WorkSpace, they have the  icon. They may be treated as native *.r2s files.
- **MapInfo Interchange files - *.mif** - Only the GIS features in point or line format, 2D or 3D, are supported by EnSim. The legends and other view decorations cannot be imported. Shape files are represented in the WorkSpace by the MapInfo icon: . The data they contain, lines or points, are represented as children with an icon identifying the specific data type. Each data type may have multiple attributes. MapInfo files can possess multiple children, up to one per type of data.

When a MapInfo Interchange file is opened in EnSim, it may be treated as a line set, point set, XYZ point set, or parcel set, depending on the type of data it contains.

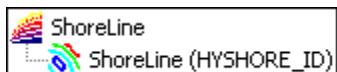


Figure A.5: A MapInfo Interchange file in the WorkSpace

- **Surfer Grid files - *.grd** - Surfer grids are loaded into EnSim as rectangular grids. In the WorkSpace, they appear with the  icon. They may be treated as native *.r2s files.
- **Binary Rasters (ARC/BIL/GTOPO) - *.hdr, *.dem, *.bil** - ArcInfo rasters, GTOPO30's and BIL files are loaded into EnSim as rectangular grid files. They are represented in the WorkSpace by the  icon. They may be treated as native *.r2s files.

- **SRTM Grid file - *.hgt** - Shuttle Radar Topography Mission files, both one-arc second (SRTM1) and three-arc second (SRTM3) are loaded into EnSim as rectangular grid files. They are represented in the WorkSpace by the  (for SRTM1 files) or  (for SRTM3 files) icons. They may be treated as native *.r2s files.
- **GeoTIFFs - *.tif** - GeoTIFFs are georeferenced images, and may be displayed in 2D or 3D views. Non-georeferenced tiffs may also be loaded and can be manually georeferenced in that the corners of the image can be assigned coordinates. GeoTIFFs can be saved as .r2s or .xyz files. They are represented in the WorkSpace by the  icon.
- **WMO GRIB Files - *.grb** - GRIB (GRIdded Binary) files are used to store meteorological data. EnSim supports Version 1 GRIB files, the sub-contents of which are treated as 2D rectangular grid files. GRIB files are represented in the WorkSpace by the  icon.

WMO GRIB Files each contain a single timestep of a dataset. You can load multiple timesteps as a single R2S file by selecting **File**→**Import**→**WMO GRIB Files**→**Multiple GRIB Files** from the menu bar. Select the files and click Open. You'll then be prompted to save the GRIB files as a single R2S file, which is loaded automatically. The files will be sorted sequentially according to the Canadian Meteorological Centre naming convention for GRIB files, which is detailed on the Environment Canada website at this location:
http://www.weatheroffice.gc.ca/grib/Ensemble_GRIB_e.html.

GeoTIFF Theme files [thm]

EnSim provides an ASCII theme file (*.thm) to support classification of GeoTIFF images (see "Classification of a GeoTIFF Image", on p. 130. The theme file contains the categorization of pixel values to colours and class names. The theme file is an EnSim file and can be loaded and saved from the **Classes** tab of the **GeoTIFF Properties** dialog. These files are stored in the *Application\bin\Templates\GEOTIFF* directory.

An example of an ASCII theme file is shown below.

```
# CLASSIFICATION THEME FILE
# Original name EC
# INDEX COLOUR TEXT
:CLASS 21 0x2222b1 Impervious
:CLASS 22 0x00fbff Deciduous
:CLASS 23 0x006400 Coniferous
:CLASS 24 0x238d6c Agriculture
:CLASS 25 0x20a2d9 Pasture
:CLASS 26 0x254283 Wetland
:CLASS 27 0xeeeeeaf Water
```

- **:CLASS** - Each class is defined by the pixel value or index, the RGB colour value, and the class name.

APPENDIX B: FILE TYPES OF GREEN KENUE

Green Kenuue is capable of reading all of the file types that are either native to or supported by EnSim Core. In addition, Green Kenuue has one additional native file type—the watershed object—and supports twelve more external file types. This extra file type allows Green Kenuue to be an effective integrated numerical modelling environment for hydrological models.

For EnSim conventions regarding file headers and common keywords, see "File Headers", on p. 264.

NATIVE FILE TYPES

Watershed Objects [wsd]

The watershed object is the most important file type in Green Kenu, since it contains the basic geographical and geophysical data necessary to create the hydrological model. All data values in the watershed object correspond to a vertex of the DEM, which forms the basis of the watershed object.

The file extension for a watershed object is ***.wsd**, and it is represented in the WorkSpace by the  icon. The three data objects that are contained within a watershed objects are also represented by icons. The DEM has the icon of a 2D scalar rectangular grid: ; the channel object has the network icon: ; and the Basin has a unique icon: . The basin object otherwise has the properties of a 3D line set.

File Headers [wsd]

An example of a watershed object file header is shown below.

```
#####
:FileType wsd  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType          WaterShed
#
:Application        GreenKenu
:Version            3.1.33
:WrittenBy          Username
:CreationDate       Fri, Apr 15, 2005 11:20 AM
#
#-----
#
:Name   WaterShed
#
:Projection LatLong
:Ellipsoid NAD80
#
:AttributeName 1 Elevation
:AttributeName 2 Direction
#
:FlowAlgorithm      AT_Search
#
:HydroGrid
  :xOrigin  409000.000000
  :yOrigin  4978000.000000
  :xCount   101
  :yCount   101
  :xDelta   400.000000000000
  :yDelta   400.000000000000
  :Angle    0.000000
  :SourceFile   JockRiver.r2s
  :Name DEM
:EndHydroGrid
#
:HydroChannels
```

```

:DrainageAreaThreshold 8.000000
:MinWaterShedArea 375.000000
:MinAdjWaterShedArea 26.000000
:OutletsViewable 0
:Title Network
:Name Channels
:EndHydroChannels
#
:Basin 1
:Outlet_x 445400.000000
:Outlet_y 5017600.000000
:Title Basin
:Name Basin 1
:EndBasin
#

```

For an explanation of general keywords used in file headers, see "File Headers", on p. 264. The **:Name** and **:Title** keywords that are used for several watershed subcomponents are described in that section.

The **:flowAlgorithm** keyword indicates which algorithm was used to generate the depressionless DEM from the source data.

The Watershed object uses several keywords that are similar to those found in a 2D rectangular grid to describe the depressionless DEM. These keywords are found after the keyword **:HydroGrid**, which indicates the beginning of the subsection and before the **:EndHydroGrid** keyword, which indicates its end:

- **:xOrigin** - This is the x-coordinate of the point in the bottom left corner of the grid.
- **:yOrigin** - This is the y-coordinate of the point in the bottom left corner of the grid.
- **:xCount** - The number of points, or vertices, in each row of the grid, along the x-direction.
- **:yCount** - The number of points, or vertices, in each column of the grid, along the y-direction.
- **:xDelta** - The distance between two adjacent points in a row.
- **:yDelta** - The distance between two adjacent points in a column.
- **:Angle** - The clockwise angle of rotation, in degrees, of the grid about the origin, or bottom-left corner.

There is one additional keyword that is not found in a 2D rectangular grid

- **:SourceFile** - The name of the file from which the Depressionless DEM was generated.

Several keywords are used to describe the channels. These keywords are enclosed by the **:Hydrochannels** and **:EndHydroChannels** keywords.

- **:DrainageAreaThreshold** - The minimum drainage area required to treat a flow path as a channel that is displayed in a view within EnSim. Channels displayed in a view will have an upstream drainage area equal to or greater than this value.

- **:MinWaterShedArea** - The minimum upstream drainage area of a point along a channel for that point to be considered as the outlet of a watershed.
- **:MinAdjWaterShedArea** - Watersheds adjacent to the potential watershed outlets must have upstream drainage areas equal to or greater than this value.
- **:OutletsViewable** - Specifies whether the target watershed outlets are displayed within a view. 1 indicates yes, and 0 indicates no. Target outlet nodes are based in the criteria specified by the **:MinWaterShedArea** and **:MinAdjWaterShedArea** keywords.

The following keywords are specific to the Basin portion of the watershed object, and are enclosed by the **:Basin** and **:EndBasin** keywords.

- **:Outlet_x** - The x-coordinate of the outlet of this watershed basin.
- **:Outlet_y** - The y-coordinate of the outlet of this watershed basin.

File Format [wsd]

ASCII

The watershed object file contains three different types of information:

- The Depressionless DEM, in the form of a 2D rectangular grid
- A flow direction grid, also in the form of a 2D rectangular grid
- Basin data, which is written in the file header

The data section of the file is divided into two sections, both similar to 2D rectangular grids. For information on these sections, see "2D Rectangular Grids [r2s / r2v]", on p. 270.

The values of the data in the first section of the watershed object are elevations derived from the DEM. The second section pertains to the flow of water in the watershed. For each vertex, there is an integer between 1 and 8. The integer values correspond with a compass direction, which indicates the direction of flow.

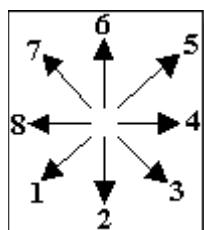


Figure B.1: These values are found in the second section of the watershed object

Binary

In a binary watershed, all header information remains in ASCII format. The data sections are written in binary format. The first section, the Depressionless DEM, is stored as a series of 4-byte floating point values, each representing an elevation. The second section, the flow directions, is stored as a series of 1-byte characters. Both sections use the same data order as the ASCII version of the file type.

SUPPORTED FOREIGN FILE TYPES [GREEN KENUE]

Green Kenue supports all of the file types supported by EnSim Core. In addition to those files, Green Kenue supports the following file types.

- **Watflood map file - *.map - ** - Map files are displayed as r2c grids, but the file format is quite different. See your WATFLOOD documentation for the format of map files. Map files may be created, edited, and saved with Green Kenue.
- **Watflood output file - *.wfo - ** - Format designed for WATFLOOD by CHC. This is the output file from a WATFLOOD hydrologic model. The components of a Watflood output file are in binary r2c format. Each component of the wfo file can be saved individually as either a single-time-frame ASCII r2c file, or as a multiple-time-frame binary r2c file. See the file format for "2D Rectangular Cell Grids [r2c]", on p. 300, for more information.
- **Watflood Met file - *.met - ** - Distributed rain data. Components are stored as r2c objects.
- **Watflood Rad file - *.rad - ** - Radar data. Components are stored as r2c objects.
- **Watflood Rag file - *.rag - ** - Rain gauge data. Components are stored as time series data objects.
- **Watflood Rel file - *.rel - ** - Reservoir release data. Components are stored as time series data objects.
- **Watflood Shd file - *.shd - ** - WATFLOOD basin data. Components are stored as r2c objects.
- **Watflood Snw file - *.snw - ** - Snow data. Components are stored as time series data objects.
- **Watflood Spl Plt file - *.plt - ** - Computed stream flow data. Components are stored as time series data objects.
- **Watflood Str file - *.str - ** - Stream flow data. Components are stored as time series data objects.
- **Watflood Tag file - *.tag - ** - Temperature gauge data. Components are stored as time series data objects.
- **Watflood Tem file - *.tem - ** - Distributed temperature data. Components are stored as r2c objects.
- **Environment Canada RPN Files - *.fst** - A generic compressed data file format used to distribute Wind Energy data files, among other data. This is a native file type for the AnemoScope program, available at <http://www.anemoscope.ca/>. Further information on the *.fst file type can be found at <http://www.windatlas.ca/en/fst.php>.
- **Topaz watershed file** - See Topaz documentation for information on the format of these files. Only Topaz output files in ArcInfo ASCII grid file (*.arc) format are supported. Topaz watershed files cannot be created, edited, or saved with Green Kenue. They may be loaded into Green Kenue and displayed in a 2D or 3D view, or may be used to create a new EnSim watershed object.

- **URP Metafile** - Unified Radar Processor is the weather radar format used by Environment Canada. URP Metafiles are loaded into EnSim as triangular meshes. In the WorkSpace, they have the icon . They may be treated as native *.t3s files.

APPENDIX C: FILE TYPES OF THE RCA

The Rating Curve Analysis File [rca]

The file extension for a **Rating Curve Analysis** object is *.rca. It is represented in the WorkSpace by the icon. For information about using rating curve analyses, see "Rating Curve Analysis (RCA)", on p. 165.

File Header [rca]

An example of a header from an RCA object is shown below. This RCA was generated with the Power Curve function.

```
#####
:FileType rca ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType Rating Curve Analysis
#
:Application GreenKenue
:Version 3.1.33
:WrittenBy Username
:CreationDate Fri, Apr 15, 2005 11:20 AM
#
#-----
:#Name RatingCurveAnalysis 03BC001
#
:DischargeSourceName Flow03BC001
:LevelSourceName Level03BC001
#
# The rating curve has been fitted in the form:
# Power Function Q = C * (H - H0) ^ N
# C : Constant numerically equal to discharge when (H-H0) = 1.0
# N : Slope of the rating curve
# H0 : Effective gauge height of zero flow (also called Scale offset).
#
:PowerCoeffLogC -62.194
:PowerCoeffN 20.4427
:PowerCoeffH0 0
#
```

The explanation of the general keywords used in the header is in Appendix A, in the section "File Headers", on p. 264. Keywords specific to the RCA are discussed below.

- **:DischargeSourceName** - This is the name of the discharge, or flow object used for the RCA.
- **:LevelSourceName** - This is the name of the level, or stage object used for the RCA.

Below the Source identifiers, there is a description of the rating curve type used. If the RCA was generated with the Power Curve function, the following keywords are used:

- **:PowerCoeffLogC** - The Log C coefficient value.
- **:PowerCoeffN** - The slope of the rating curve coefficient.

- **:PowerCoeffHO** - The effective gauge height at zero flow.

If the RCA was generated using the Polynomial function, the following lines will replace those describing the Power Curve coefficients:

```
# The rating curve has been fitted in the form:  
# Polynomial fit of the square root of the discharge.  
#  $\text{sqrt}(Q) = A + B*H + C*H^2 + D*H^3 + E*H^4 + F*H^5 + G*H^6$ .  
#  
:.SqrtCoeffA      -340.031  
:.SqrtCoeffB      12.6519  
#
```

If the RCA was generated using the Polynomial function, the following keywords are used:

- **:.SqrtCoeff[A|B|C|D|E|F|G]** - This keyword defines the square root coefficient for the polynomial. **:.SqrtCoeffA** and **:.SqrtCoeffB** are present for a line, while additional keywords—**:.SqrtCoeffC** through **:.SqrtCoeffG**—are included for polynomials of greater order.

File Format [rca]

ASCII

All RCA objects are saved in ASCII format. Only the flow and the corresponding discharge for each record used in the RCA are saved within the file.

The following is an example of three records from the file whose header is excerpted above:

```
1963/07/20 00:00:00.000 847.000000 29.059999 0  
1963/07/21 00:00:00.000 841.000000 29.049999 0  
1963/07/22 00:00:00.000 813.000000 29.000000 0
```

Binary

Because they do not vary over time, there is no binary file format for RCA files.

APPENDIX D: FILE TYPES OF GEN1D

The GEN1D Parameter File

The GEN1D parameter file, ***.g1d**, is the only native file type for this model within Green Kenu. This parameter file controls all aspects of a GEN1D simulation.

File Header [g1d]

The header of a GEN1D parameter file consists of only the general EnSim Header shown below. For an explanation of the keywords used, see "File Headers", on p. 264.

```
#####
:FileType g1d  ASCII  EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType          GEN1D Parameter Set
#
:Application        GreenKenu
:Version            3.1.33
:WrittenBy          Username
:CreationDate      Fri, Apr 15, 2005 11:20 AM
#
#-----
#
```

File Format [g1d]

The contents of an example GEN1D parameter file are shown below.

```
# Simulation
#
:NAME          GEN1DSIM
#
:RUN_TYPE      RUN
:DELTA_T       0:00:10.000
:SIMULATION_TIME 3:55:00
#
:VARY_DELTA_T TRUE
:COURANT       1.000000
#
:VISCOSITY     1.000000
#
# Input Files
#
:CHANNEL_NETWORK_FILE GEN1D-Example.n3s
#
:DOWN_BOUNDARY_NODE_ID 1
:DOWN_BOUNDARY_TYPE   FREE_FLOW
:DOWN_BOUNDARY_VALUE  0.000000
#
:UP_BOUNDARY_NODE_ID 18
:UP_BOUNDARY_TYPE   LEVEL_CONSTANT
:UP_BOUNDARY_VALUE  0.000000
```

```

:STEADY_STATE_DISCHARGE_ACCURACY 0.0001
:CAL_FRICTION_NODE_ID 1
:CAL_FRICTION_WATER_LEVEL 0.000000
:CAL_FRICTION_MIN_STRICKLER 10.000000
:CAL_FRICTION_MAX_STRICKLER 50.000000
:RATING_CURVE_NODE_ID 1
:RATING_CURVE_DISCHARGE_START 2.000000
:RATING_CURVE_DISCHARGE_DELTA 1.000000
:RATING_CURVE_STEP_COUNT 10
#
# Output
#
:RESULT_FILE GEN1D2
:SAVE_WIDTH True
:SAVE_AREA True
#
# End of File

```

Simulation Parameters

General Parameters

- **:NAME** - Required
 - Type - text, up to 256 characters in length
 - Valid values - any valid filename without an extension
 - Default - none
 - Description - the name of the run. GEN1D uses this parameter as the basis for generated output filenames.

Simulation Parameters

- **:RUNTYPE** - Required
 - Type - text
 - Valid values - RUN, RUN_TO_STEADY_STATE, RUN_CAL_FRICTION, RUN_GEN_RATING_CURVE
 - Default - RUN
 - Description - records the type of run that has been saved. This parameter also affects which other parameters are included in the file.
- **:DELTA_T** - Required
 - Type - floating point, in seconds; or hhhh:mm:ss, in hours:minutes:seconds, separated by colons.
 - Valid values - any positive number
 - Default - none
 - Description - defines the simulation time step. This value is very dependent on the hydrodynamics used.

- **:VARY_DELTA_T** - Required
 - Type - Boolean
 - Valid values - TRUE, FALSE
 - Default - FALSE
 - Description - Records whether the simulation can vary the time step as needed.
- **:SIMULATION_TIME** - Required
 - Type - floating point, in seconds; or hhhh:mm:ss, in hours:minutes:seconds, separated by colons.
 - Valid values - any positive number
 - Default - none
 - Description - defines the length of a simulation.
- **:START_TIME** - Optional
 - Type - floating point, in seconds; or hhhh:mm:ss, in hours:minutes:seconds, separated by colons.
 - Valid values - any positive number
 - Default - none
 - Description - identifies the start time of the simulation, if it has an absolute time.

Constants

- **:COURANT** - Optional
 - Type - floating point
 - Valid Values - between 0 and 1
 - Default - 1.000000
 - Description - Records the Courant number, in the case of a variable time step. The Courant number is the ratio of the physical speed of the model to its calculation speed.
- **:VISCOSITY** - Optional
 - Type - floating point
 - Valid Values - greater than 0
 - Default - 1.000000
 - Description - Records the viscosity value of the water in the simulation. Water is considered to have a viscosity of 1.
- **:STEADY_STATE_DISCHARGE_ACCURACY** - Optional
 - Type - floating point
 - Valid Values - greater than 0 and less than 1. Invalid values are considered to be 1e-07.

- Default - 0.0001
- Description - Records the accuracy to which the simulation will be run, if the run type is `RUN_TO_STEADY_STATE`.
- **:CAL_FRICTION_NODE_ID** - Optional
 - Type - integer
 - Valid Values - between 0 and the maximum number of nodes.
 - Default - none
 - Description - records the node location of the measured node for a run type of `RUN_CAL_FRICTION`.
- **:CAL_FRICTION_NODE_LEVEL** - Optional
 - Type - floating point
 - Valid Values - greater than 0
 - Default - none
 - Description - Records the water level of the node identified in `CAL_FRICTION_NODE_ID`, for a `RUN_CAL_FRICTION` run type.
- **:CAL_FRICTION_MIN_STRICKLER** - Optional
 - Type - floating point
 - Valid Values - greater than 0
 - Default - 10
 - Description - the estimated lower limit of possible Strickler values for a `RUN_CAL_FRICTION` run.
- **:CAL_FRICTION_MAX_STRICKLER** - Optional
 - Type - floating point
 - Valid Values - greater than 0
 - Default - 50
 - Description - the estimated upper limit of possible Strickler values for a `RUN_CAL_FRICTION` run.
- **:RATING_CURVE_NODE_ID** - Optional
 - Type - integer
 - Valid Values - between 0 and the maximum number of nodes.
 - Default - none
 - Description - records the node location to be monitored to generate the rating curve in a run type `RUN_GEN_RATING_CURVE`.
- **:RATING_CURVE_DISCHARGE_START** - Optional

- Type - floating point
- Valid Values - greater than 0
- Default - 2.000000
- Description - Records the initial discharge value at which the rating curve is generated.
- **:RATING_CURVE_DISCHARGE_DELTA** - Optional
 - Type - floating point
 - Valid Values - greater than 0
 - Default - 1.000000
 - Description - Records the change in discharge between values of the rating curve.
- **:RATING_CURVE_STEP_COUNT** - Optional
 - Type - integer
 - Valid Values - any integer equal to or greater than 1.
 - Default - 10
 - Description - Records the number of generated values used to generate a rating curve in a RUN_GEN_RATING_CURVE run.

Input Files

- **:CHANNEL_NETWORK_FILE** - Required
 - Type - text
 - Valid Values - the name of any existing network file
 - Default - none
 - Description - the name of the n3s file that contains the channel network.

Boundaries

- **:DOWN_BOUNDARY_NODE_ID** - Required
 - Type - integer
 - Valid Values - between 0 and the maximum number of nodes.
 - Default - none
 - Description - records the node location of the downstream boundary.
- **:DOWN_BOUNDARY_TYPE** - Required
 - Type - text
 - Valid Values - LEVEL_CONSTANT, LEVEL_SERIES, DISCHARGE_CONSTANT, DISCHARGE_SERIES, FREE_FLOW
 - Default - LEVEL_CONSTANT

- Description - records the type of the downstream boundary node.
- **:DOWN_BOUNDARY_VALUE** - Required for LEVEL_CONSTANT or DISCHARGE_CONSTANT downstream boundary types.
 - Type - floating point
 - Valid Values - any positive number.
 - Default - none
 - Description - records the value of the downstream boundary level or discharge.
- **:DOWN_BOUNDARY_FILE** - Required for LEVEL_SERIES or DISCHARGE_SERIES downstream boundary types.
 - Type - text
 - Valid Values - the name of any existing Type 1 time series file.
 - Default - none
 - Description - records the name of the file containing varying water levels for some boundary types.
- **:UP_BOUNDARY_NODE_ID** - Required
 - Type - integer
 - Valid Values - between 0 and the maximum number of nodes.
 - Default - none
 - Description - records the node location of the upstream boundary.
- **:UP_BOUNDARY_TYPE** - Required
 - Type - text
 - Valid Values - LEVEL_CONSTANT, LEVEL_SERIES, DISCHARGE_CONSTANT, DISCHARGE_SERIES, REFLECTIVE
 - Default - LEVEL_CONSTANT
 - Description - records the type of the upstream boundary node.
- **:UP_BOUNDARY_VALUE** - Required for LEVEL_CONSTANT or DISCHARGE_CONSTANT upstream boundary types.
 - Type - floating point
 - Valid Values - any positive number.
 - Default - none
 - Description - records the value of the upstream boundary level or discharge.
- **:UP_BOUNDARY_FILE** - Required for LEVEL_SERIES or DISCHARGE_SERIES upstream boundary types.
 - Type - text

- Valid Values - the name of any existing Type 1 time series file.
- Default - none
- Description - records the name of the file containing varying water levels for some boundary types.

Output

- **:SAVE_WIDTH** - Optional
 - Type - Boolean
 - Valid values - TRUE, FALSE
 - Default - FALSE
 - Description - Records whether the width values will be saved in the output file.
- **:SAVE_AREA** - Optional
 - Type - Boolean
 - Valid values - TRUE, FALSE
 - Default - FALSE
 - Description - Records whether the area values will be saved in the output file.
- **:RESULT_FILE** -
 - Type - text
 - Valid Values - the name of any existing network file
 - Default - none
 - Description - records the name of the output of the simulation. If this field does not exist, EnSim will create it when any output of the simulation is first generated.

APPENDIX E: FILE TYPES OF HBV-EC

The HBV-EC model has three unique filetypes, ***.hbv**, which contains all of the parameters required to run the simulation, ***.hbm**, which contains meteorological data, and ***.hbt**, the HBV-EC output tableset.

The HBV-EC Parameter Set File

The .hbv file type has two variants: the default **HBV-EC Parameter Set (ASCII)**, as described below, and the **HBV-EC Parameter Set (ASCII - Without Spatial Data)**, which excludes the spatial data. This allows for a substantially smaller file size, and can be used to run the HBV-EC model from the command line with the spatial data, located in external files, supplied as arguments. Either file type can be selected from the **Save as type** drop-down menu on the **Save As** dialog.

File Header [hbv]

The header of an HBV-EC Parameter Set file consists of a listing of the objects contained within the **HBV-EC Parameter Set** object, as well as the values used to define them.

```
#####
:FileType hbv ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType          HBV-EC Parameter Set
#
:Application        GreenKenuue
:Version            3.1.33
:WrittenBy          Username
:CreationDate       Fri, Dec 16, 2005 03:23 PM
#
#-----
#
:Projection Cartesian
:Ellipsoid Sphere
#
:SourceWaterShedFile SampleWaterShed.wsd
#
:AttributeName 1 Elevation
:AttributeName 2 Direction
#
:FlowAlgorithm      AT_Search
#
:HydroGrid
  :xOrigin  481900.000000
  :yOrigin  5462000.000000
  :xCount   360
  :yCount   640
  :xDelta   50.000000000000
  :yDelta   50.000000000000
  :Angle    0.000000
  :Name     SampleDEM
:EndHydroGrid
```

```

#
:HydroChannels
  :DrainageAreaThreshold 2.000000
  :MinWaterShedArea 150.000000
  :MinAdjWaterShedArea 10.000000
  :OutletsViewable 0
  :Title Network
  :Name Channels
:EndHydroChannels
#
:Basin 1
  :Name Basin 1
  :Outlet_x 489700.000000
  :Outlet_y 5463100.000000
  :Title Basin
  :Name Basin 1
:EndBasin
#
:HBVEC 1
  :Spatial
    :NodeCount 86029
    :Elevations
      :Levels 4 600 900 1305.2
    :EndElevations
    :Slopes
      :Levels 2 25
    :EndSlopes
    :Aspects
      :Type 2
    :EndAspects
  :EndSpatial
  :Model
    :WaterShed
      :Name SampleWatershed
      :ModelTimeStep 24
      :StartDate 1996/10/01 00:00
      :EndDate 1997/09/30 00:00
      :RoutingModel Parallel
      :RunoffKF 0.4
      :RunoffAlpha 0.3
      :RunoffKS 0.1
      :RunoffFRAC 0
      :InitialFastReservoirDischarge 10
      :InitialSlowReservoirDischarge 10
      :InitialFastReservoirTemperature 8
      :InitialSlowReservoirTemperature 8
      :OutletElevation 0
      :ClimateZone
        :Name Climate Zone 1
        :METFile YVR_1Year.hbm
        :AtmosphereRFCF 1
        :AtmosphereSFCF 1
        :AtmospherePGRADL 0.0005
        :AtmospherePGRADH 0
        :AtmosphereEMID 5000
        :AtmosphereTLAPSE 0.005
        :AtmosphereTT 0
        :AtmosphereTTI 1
        :AtmosphereEPGRAD 0.0005

```

```

:AtmosphereETF 0.5
:ForestTFRAIN 0.8
:ForestTFSNOW 0.7
:ForestCanopyFactor 1.0
:SnowAM 0.05
:SnowTM 0
:SnowCMIN 3
:SnowDC 2.5
:SnowMRF 0.9
:SnowCRFR 2
:SnowWHC 0.5
:SnowLWR 2500
:SoilFC 200
:SoilBETA 0.7
:SoilLP 0.8
:GlacierMRG 1.5
:GlacierAG 0.5
:GlacierDKG 0.2
:GlacierKGMin 0.1
:GlacierKGRC 0.7
:ElevationBand
  :Name Elevation Band 1
  :Elevation 435.100000
  :LandUse Forest
    :Name Forest (slope 12.5) (aspect 0)
    :Aspect 0.000000
    :Slope 15.000000
    :InitialSnowSolid 0.000000
    :InitialSnowLiquid 0.000000
    :InitialSoilMoisture 0.000000
    :InitialSoilWaterTemperature 0.000000
  :EndLandUse
  :LandUse Forest
    :Name Forest (slope 12.5) (aspect 90)

```

[...Other Land Use Descriptions...]

```

  :EndLandUse
:EndElevationBand
:Elevation Band
  :Name Elevation Band 2

```

[...Other Elevation Bands...]

```

  :EndElevationBand
:EndClimateZone
:ClimateZone
  :Name Climate Zone 2

```

[...Other Climate Zones...]

```

  :EndClimateZone
  :EndWaterShed
  :EndModel
:EndHBVEC
:EndHeader

```

For an explanation of general keywords used in file headers, see "File Headers", on p. 264. The **:Name** and **:Title** keywords that are used for several subcomponents are described in that section.

For a detailed description of the parameters used to define the Watershed object, see "Watershed Objects [wsd]", on p. 308. Within the ***.hbv** file shown below, the Watershed parameters appear between the **:HydroGrid** and **:EndBasin** keywords.

The keywords found in the **:Model** section of the header are the same as the variable names used in the respective panels of the **Simulation** and **Climate Zone** panels of the **HBV-EC Parameter Set** object. See "The Simulation Panel", on p. 250, and "The Climate Zone Panel", on p. 253 for details on the meanings of the keywords.

File Format [hbv]

HBV files are always stored in ASCII format. The body of the file consists of a list of values for the parameters identified by the **:AttributeName** keywords. In general, this will be the **Elevation** and **Flow Direction** of each node in the watershed, starting in the bottom-left corner and going from left to right, top to bottom.

The HBV-EC HBM File

The HBM file, formerly known as the MET file, is used to supply recorded meteorological data to the HBV-EC simulation.

File Header [hbm]

The header of an HBV-EC HBM meteorological data file consists of a set of keywords that describe the structure of the subsequent data.

```
#####
:FileType hbm ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2005
# DataType HBV-EC Hbm
#
:Application           GreenKenue
:Version               3.1.33
:WrittenBy             Username
:CreationDate          Wed, Jan 25, 2005 03:23 PM
#
#-----
#
#
#:ColumnMetaData
#:ColumnName Rainfall Snowfall Temperature
#:ColumnUnits mm mm degC
#:ColumnType float float float
#:EndColumnMetaData
#
:StationName  Sample Weather Station
:Elevation    430.000000
:LocationX   49.57
:LocationY   -119.23
#
:MonthlyTemperature -2.0 -2.7 3.8 7.5 12.6 14.7 19.3 19.6 14.4 6.3 3.9 -1.0
:MonthlyEvaporation 8 4 10 25 19 32 34 16 18 23 19 21
#
:StartTime   1988/10/01 00:00
:DeltaT     24:00:00.000
:EndHeader
0 0 17.3
0 0 14.3
0 0 14.5
7.6 0 12.5
0 0 11.5
0 0 11.8
0 0 11.8
0 0 12.5
0 0 15
0 0 14
0 0 12.5
2 0 12.8
18 0 12.8
29.3 0 12
```

For an explanation of general keywords used in file headers, see "File Headers", on p. 264.

- **:ColumnMetaData** - Required - This keyword marks the start of the metadata information for the body of the file.
- **:ColumnName**, **:ColumnUnits**, **:ColumnType** - Required - These keywords are followed by the Names, Units, and Types, respectively, of each of the columns that appear in the body of the file. The values must be listed in the same order as the columns appear.

Note: The units for measurement of snowfall is given in millimetres of water equivalent, not centimetres of snowfall. In general, the water equivalent is considered to be one tenth of the measured depth of fresh snow.

- **:EndColumnMetaData** - Required - This keyword marks the end of the metadata information.
- **:StationName** - Optional - This keyword is followed by the name of the weather station.
- **:Elevation** - Required - This keyword is followed by the elevation of the weather station at which the data was recorded, in metres.
- **:LocationX** and **:LocationY** - Required - These values give the location of the weather station, in the units of the source material. Since these values are not actually used by the HBV-EC model, they are included for reference purposes only, to help you locate a weather station within (or near) a watershed. If they are not included in the HBM file, they're assumed to be 0, the default value.
- **:MonthlyTemperature** - Required - This keyword is followed by the average monthly temperatures recorded for the weather station for each month of the year, in order, in degrees Celsius.
- **:MonthlyEvaporation** - Required - This keyword is followed by the average monthly evaporation amount recorded for the weather station for each month of the year, in order, in millimetres.
- **:StartTime** - Required - This keyword is followed by the time of the first measurement listed in the body of the file, in the format YYYY/MM/DD HH:mm.
- **:DeltaT** - Required - This keyword is followed by the interval between the measurements listed in the body of the file, in the format HH:mm:ss.sss.

File Format [hbm]

HBM files are always stored in ASCII format. The body of the file consists of columns of data. The columns are as described by the **:ColumnMetaData** keywords, and each row represents a single measurement at a particular weather station, separated by the time interval given by the **:DeltaT** keyword.

The HBV-EC HBT File

The HBV-EC HBT TableSet is an alternate form of output for the HBV-EC model. Its creation is triggered by activating the **Generate Table File** option in the **Output Options** section of the **Simulation** panel of the HBV-EC **Properties** dialog. It consists largely of a series of nested tables, similar in format to the EnSim Core .tb0 object, which is detailed in "Tables [tb0]", on p. 293

File Header [hbt]

```
#####
:FileType hbt ASCII EnSim 1.0
# Canadian Hydraulics Centre/National Research Council (c) 1998-2009
# DataType HBVEC Output TableSet
#
:Application           GreenKenu
:WrittenBy             Username
:CreationDate          Wed, Apr 14, 2010 03:23 PM
#
#-----
#
:Name                  Sample HBT File
#
:Table
:Name                  Sample HBT File
:Level                 Watershed
:StartTime              1983/12/31 00:00
:EndTime                2000/12/31 00:00
:DeltaTimeHours         24
:Area                  273.735138
:LandArea               263.175911
:LakeArea                1.196895
:GlacierArea             9.362332
:ColumnMetaData
   :ColumnName TotalDischarge FastReservoirDischarge SlowReservoirDischarge
   GlacierReservoirDischarge FastReservoirStorage SlowReservoirStorage Rainfall
   Snowfall PrecipLand PrecipLake PrecipGlacier InterceptionLand SnowWaterEquivalent
   SWELand SWEGlacier SoilInfiltration SoilInfiltrationLand SoilMoisture
   SoilMoistureLand UnitEvaporation EvaporationLand EvaporationLake WaterRelease
   WaterReleaseLandFast WaterReleasePercSlow WaterReleaseGlaciers GlacierIceMelt
   :ColumnUnits m3/s m3/s m3/s mm mm
   mm mm mm mm mm mm
   :ColumnType float float float float float float float float float float
   float float float float float float float float float float float float
   float float
   :EndColumnMetaData
   :EndHeader
```

...data columns describing the contents of the Watershed...]

...similar tables for each of the terrain types nested by Climate Zone, Elevation Band, and Land Class (a combination of Terrain Type, Slope, and Aspect)...

For an explanation of general keywords used in file headers, see "File Headers", on p. 264. An explanation of keywords specific to or prominent in HBT files follows.

- **:Level** - Required - This keyword indicates the nesting level of the subsequent table. From highest level to lowest, the options are `Watershed`, `Zone`, `Band`, or `Class`, corresponding to the entire watershed, the climate zone, the elevation band, or a particular combination of terrain type, slope, and elevation, respectively..
- **:Type** - Required for `Class`-level tables - This keyword indicates the terrain type of the described class. It may be `Open`, `Forest`, `Glacier`, or `Lake`.
- **:Area** - Required - This keyword indicates the total area, in square kilometers, described by the given table and its nested tables.
- **:LandArea** - Required, for `Watershed`, `Zone`, or `Band` tables - This keyword indicates the total area, in square kilometers, of the land found within the given watershed, climate zone, or elevation band. This is the sum of the `Open` and `Forest` terrain types.
- **:LakeArea** - Required, for `Watershed`, `Zone`, or `Band` tables - This keyword indicates the total area, in square kilometers, of lake found within the given watershed, climate zone, or elevation band. Note that `Lake` terrain always has a slope and aspect of zero, so all lake terrain found within a given elevation band is contained in a single `Class` table.
- **:GlacierArea** - Required, for `Watershed`, `Zone`, or `Band` tables - This keyword indicates the total area, in square kilometers, of glacier found within the given watershed, climate zone, or elevation band.
- **:Elevation** - Required for `Band`- or `Class`-level tables - This keyword indicates the elevation, in metres, of the described terrain.
- **:ColumnMetaData** - Required - This keyword marks the start of the metadata information for the body of the file.
- **:ColumnName**, **:ColumnUnits**, **:ColumnType** - Required - These keywords are followed by the Names, Units, and Types, respectively, of each of the columns that appear in the body of the subtable. The values must be listed in the same order as the columns appear.
- **:EndColumnMetaData** - Required - This keyword marks the end of the metadata information.

File Format [hbt]

HBT files are always stored in ASCII format. The body of the file consists of columns of data. The columns are as described by the `:ColumnMetaData` keywords, and each row represents a single set of HBV-EC results for a particular terrain class, separated by the time interval given by the `:DeltaT` keyword.

Index

A

Anemoscope	311
Animation	61
clock	58
flight paths	62
recording	65
ArcInfo ASCII grid file	304
ArcINFO ASCII Grid files	270
ArcView Shape File	304
saving	16
Areas and Volumes	88
Aspects	92
Attribute Tables	100

B

Base Maps	60
Basin Network	159
Basins	149
creating	150
outlet nodes	147
removing	150
Binary Rasters	304
Blue Kenue	1
Borders	60

C

Calculators	115
data items	115
expressions	120
extract a temporal subset	127
gridded objects	116
time series	118

CDCD

accessing	203
file locations	203

loading a station	204
stations	
accessing	204
details	205
meta data	206
properties	205
time series	206
subsets	207

Channels	141
attributes	142
displaying	143
editing	145
outlet nodes	147
predefined	145

CHC (Canadian Hydraulics Centre) 1	
---	--

Chézy coefficient	215
--------------------------------	-----

Clock	58
--------------------	----

Compass	58
----------------------	----

Contours	
displaying isoline-outlined filled	125

Coordinate Systems	26
assigning projections	27
converting projections	27
criteria	29
datums	28
ellipsoids	28
in 2-dimensional views	40

Crookshank	215
-------------------------	-----

Cross-sections	
extracting	
from gridded data	123
from points and line data	124

Cumulative Sum	110
-----------------------------	-----

Curvatures	93
-------------------------	----

D

Data Items	9
adding other objects to	5

creating	68	labels	59
line set	69	legends	54
points	68	DEM s	304
regular grid	70	checking and editing	141
sloping structure	126	in watersheds	140
table object	72	Depression Fill	153
triangular mesh	71	Downstream Reach	158
displaying two features	124	Drainage Area	152
editing	73	Drainage Area Ratio Analysis	161
attributes	73	computed flow	163
lines	76	Drainage Directions	151
points	75		
resampling lines and linesets	80		
resampling time series	107		
shifting data objects	82		
T3 meshes	79		
time series	104		
extracting			
spatial subset	127	Attribute Tables	
temporal subset	127	extracting	100
importing	10	attributes	73
loading	10	lines	76
mapping	113	points	75
native	10	resampling data	80
properties	17	shifting data objects	82
removing other objects from	5	T3 meshes	79
renaming	6	time series	104
saving	11–16		
Data Objects		Ellipsoids	28
See Data items			
Data Probes		EnSim	
areas and volumes	88	about	2
live cursor	85	applications	1
live stream lines cursor	86	conventions	2
popups	83		
ruler	87	Equations	
selection info	84	continuity	215
Data, Extracting		motion	215
See Extracting Data		Navier-Stokes	215
Datums		Extracting Data	89
See Ellipsoids		isolines	95
Decorations	53	meshes	101
clock	58	edge lengths	102
compass	58	edges	102
icon	4	subsets	101
		paths	95
		points	96
		residuals	95
		surfaces	90

aspects	92
curvatures	93
slopes	91
temporal statistics	90
time series	96
velocity rose	99
XY data	103
Extracting data	
integrals	103

F

File Headers	264
File Menu	2
File Types	
file formats	268
ASCII	268
binary	268
file headers	264
keywords	264
supported	263
Flight Paths	62
properties	63
Flow Duration Curve	110
FST Files	311

G

GEN1D	135, 215
creating hot start	235
cross-sections	228
adjusting vertically	232
associating	228
copying	229
generating	232
interpolating	231
positioning	230
properties	233
removing	233
resampling	230
scaling	229
displaying output	234
equations	215

files	
file format	315
file headers	315, 323, 327
g1d file	219
geometric requirements	216
area	217
boundaries	217
channels	217
cross-section	217
elevation	217
nodes	217
width	217
parameters	219
boundaries	319
constants	317
file	315
general	316
input	319
output	321
simulation	316
properties	219
channel	223
creating	223
editing	
node	225
segment	224
opening	224
down boundary	226
simulation	219
results	220
run type	220
temporal	220
up boundary	227
running simulation	234
GeoBase	60
Geogratis	60
GeoTIFF	305
classification	130
draping onto a DEM	122
georeferencing	129
icon	4
saving	16
saving as	270
theme files	305
Getting Help	2

Getting Started	1	simulation times	251
Green Kenu	1	watershed panel	239
GRIB Files	305	watershed, creating	240
loading multiple	305	watershed, identifying basin	241
Gridded Data		watershed, importing	240
extracting cross-sections	123	model	
H		results	260
HBM Files	327	running simulation	259
body	328	references	238
header	327	snow melt factor variation	238
HBT TableSet Files	329	the HBV-EC model	237
body	330	watershed routing	238
header	329		
HBV Parameter Set	323	Help	2
body	326	HGT Files	305
header	323	Hints	122
HBV-EC	135, 237	How To	122
algorithms	237	classify a GeoTIFF image	130
background	237	create a sloping structure	126
climate zones	238	digitize from an imported image	128
file types	323	display isoline-outlined contours	125
hbm	327	display two features of an item	124
hbt	329	drap an image onto a DEM	122
hbv	323	extract a cross-section	
history	237	from gridded data	123
interface	238	from points and line data	124
basin panel	242	extract a spatial subset	127
aspect	247	extract a temporal subset	127
climate	243	georeference a GeoTIFF	129
elevation	243		
identifying zones	248	HYDAT	
land use	245		
slope	246	accessing	193
climate zone panel	253	file locations	193
climate zone parameters	253	loading a station	195
elevation band parameters	256	overview	193
land class parameters	257	stations	
met tab	257	accessing	195
new parameters	239	details	198
simulation panel	250	filtering	196
outlet elevation	252	HYDEX	199
routing	252	meta data	200

Hypsometric Curve

See Hypsographic Curve

I

Icons	3, 4
container	4
geoTIFF	4
line set	
2D	4
3D	4
network	4
point set	4
rectangular grid	
scalar	4
vector	4
table	4
time series	
scalar	4
vector	4
triangular mesh	
scalar	4
vector	4
xy object	4

Images

copying to clipboard	66
digitizing from an imported	128
printing	66
recording	65
saving and copying	65

Independent Legends

Integrals	
computing	110
extracting	103

Interface

Isoline-Outlined Filled Contours	
displaying	125

K

Kamphuis	215
Kenu	1
Green	135, 307

L

Labels	59
Legends	
Independent	56
Legends, Colour Scale	54
Line Data	
extracting cross-sections from ...	124
Line Set	279
2D	10
icon	4
3D	10
icon	4
creating	69
editing	76
file format	
ASCII	280
binary	280
file header	279
resampling	80

Lines

see Line Set

Live Cursor**Live Stream Lines Cursor****M**

MapInfo Interchange Format	304
saving	16

Mapping Objects**Menu Bar****Meteorological Data Files**

hbm	327
body	328
header	327

Multi-Tile Watersheds	137
delineating	139

N**NARR**

accessing	209, 210
National Water Data Archive	193
Networks	11, 296
file format	
ASCII	297
binary	298
file header	296
icon	4
in GEN1D	215
O	
OilSim	7
Overview	1
P	
Parcel Set	11, 283
file format	
ASCII	284
binary	284
file header	283
Performance Statistics	109
Point Set	11, 286
creating	68
file format	
ASCII	287
file header	286
icon	4
Points	
editing	75
extracting cross-sections from ...	124
Polar View	37
Prandle	215
Predefined Channels	145
Probability Distribution	111
Projections	
assigning	27
converting	27
Properties	
applying changes	31

colour scale	20
legends	54
copying	31
data attributes	21
display	17
1-dimensional views	36
2-dimensional views	42
3-dimensional	45
other	19
polar views	39
rendering	18
scale	19
shift	19
spherical view	48
flight paths	63
meta data	31
page setup	
report view	52
spatial	25
attributes	26
coordinate systems	26, 27
ellipsoids	28
viewing	5
Q	
Quick Legend	57
R	
R2C Files	300
Rating Curve Analysis (RCA)	165
background and theory	165
polynomial curve	166
power curve	165
creating	166
file	
format	313
format	
ASCII	314
binary	314
file header	313
keywords	313
interface	166
manipulating	168

views	168	Spherical View	46
manipulating	169	SRTM Grid file	305
example	170, 172	Status Bar	
opening	174	1-dimensional view	35
saving	174	2-dimensional view	41
subsets	169	3-dimensional view	43
RCA		polar view	38
See Rating Curve Analysis		report view	49
Rectangular Grid	270	spherical view	47
cell grids	300	Stream (Strahler) Order	142
file format		Stream Power	156
ASCII	301	Surfer Grid file	304
multiframe	302	Surfer Grid files	270
binary	302	Synchronizing	63
headers	300		
creating	70		
creating a sloping structure in	126		
data organization	271		
file format	272		
ASCII	272		
binary	272		
file headers	270		
scalar	10		
icon	4		
vector	10		
icon	4		
Regular Grid			
See Rectangular Grid			
Relief Potential	157		
Report View	49		
Resampling Data	80		
RPN Files	311		
Ruler	87		
S			
Selection Info	84		
Shortcut menu			
viewing	5		
Simultaneous Displays	124		
Slope Analysis	164		
Slopes	91		
Spherical View	46		
SRTM Grid file	305		
Status Bar			
1-dimensional view	35		
2-dimensional view	41		
3-dimensional view	43		
polar view	38		
report view	49		
spherical view	47		
Stream (Strahler) Order	142		
Stream Power	156		
Surfer Grid file	304		
Surfer Grid files	270		
Synchronizing	63		
T			
T3 Mesh			
See Triangular Mesh			
Tables	11, 293		
file format			
ASCII	294		
binary	294		
file header	293		
icon	4		
Telemac, EnSim	1		
Time Series	96, 288		
computing distribution	111		
cumulative sum	110		
editing	104		
file format			
ASCII	290		
type 1	290		
type 2	290		
type 3	291		
type 4	291		
type 5	292		
binary	292		
file header	288, 289		
flow duration curve	110		
integrals	110		
performance statistics	109		
resampling	107		

scalar	
explicit	11
icon	4
implicit	11
tools	104
vector	
explicit	11
icon	4
implicit	11
Tips	122
Tool Bar	8
animation	61
report view	50
Tools	68
calculators	115
create vector field	112
creating data items	68
data probes	82
extracting data	89
mapping data items	113
time series	104
compute distribution	111
cumulative sum	110
flow duration curve	110
integral	110
performance statistics	109
Topaz watershed file	311
Triangular Mesh	274
creating	71
editing	79
extracting	101
file format	275
ASCII	275
binary	276
file header	274
scalar	10
icon	4
vector	10
icon	4
Tricks	122
U	
Upslope Elevation	154
Upslope Slope	155
Upstream Network	157
URP Metafiles	312
V	
Vector Field	
creating	112
Velocity Rose	11, 295
extracting	99
file format	
ASCII	295
file header	295
Views	33
1-dimensional	34
axes	35
display properties	36
manipulating	36
status bar	35
2-dimensional	40
coordinate systems	40
display properties	42
manipulating	41
status bar	41
3-dimensional	43
display properties	45
manipulating	44
status bar	43
adding data items to	5
animation	61
Base Maps	60
creating	33
decorations	53
polar	37
coordinates	38
display properties	39
manipulating	38
status bar	38
properties	33
properties dialog	34
removing	33
removing data items from	5
report	49
manipulating	51

page setup properties	52
status bar	49
templates	53
tool bar	50
spherical	46
display properties	48
manipulating	47
status bar	47
synchronizing	63
troubleshooting	66
W	
Watershed	135
algorithms	138
At	138
Jenson	138
basins	149
channels	141
components	135, 137
creating	136
DEM ^s	140
files	308
format	
ASCII	310
binary	310
headers	308
flow paths	141
importing from Topaz	136
opening	136
outlet nodes	147
tools	151
drainage area ratio analysis ..	161
computed flow	163
known flow	162
extracting	
basin flow path distances	160
basin network	159
depression fill	153
downstream reach	158
drainage area	152
drainage directions	151
hypsographic curve	159
relief potential	157
stream power	156
upslope elevation	154
upslope slope	155
upstream network	157
wetness index	155
slope analysis	164
Watersheds	
multi-tile	137
delineating	139
WATFLOOD	135
bankfull animation	190
event files	188
importing files	187
map files	175
creating	175
data attributes	179
calculating	182
description	179
displaying	182
editing	183
land use	
editing	187
geoTIFFs	186
polygons	183
resetting	187
multiple watersheds	178
opening	175
saving	187
output	189
supported files	311
WaveSim	1, 7
Wetness Index	155
WorkSpace, The	3
actions	5
loading	6
saving	6
X	
XY Object	282
file format	282
file header	282
icon	4
XYZ Point Set	281
file format	281

file header 281